JMAG Version 12

User's Manual Solver

JSOL Corporation

Issue date of Third Edition: February 2013

■ COPYRIGHTS

Copyright © 2012-2013, JSOL Corporation All Rights Reserved.

- Information in this document is subject to change without notice.
- No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or any means electronic or mechanical, including photocopying and recording for any purpose other than the purchaser's personal use without the written permission of JSOL Corporation.

JMAG Group

Engineering Technology Division

JSOL Corporation

Tokyo/ 2-5-24, Harumi, Chuo-ku, Tokyo, 104-0053 Japan
TEL: +81(3) 5859-6020 TEL(Support): +81(3) 5859-6023 FAX: +81(3) 5859-6035

Osaka/ 2-2-4 Tosabori, Nishi-ku, Osaka, 550-0001 Japan
TEL: +81(6) 4803-5820 FAX: +81(6) 6225-3517

E-mail: jmag-support@sci.jsol.co.jp URL: http://www.jmag-international.com/

■ TRADEMARKS

- JMAG and JMAG-RT are registered trademarks of JSOL Corporation.
- Abaqus is a trademark of Dassault Systemes S.A. and Dassault Systemes SIMULIA Corp.
- AMD Opteron is a trademark of Advanced Micro Devices, Inc.
- Intel and Xeon are trademarks or registered trademarks of Intel Corporation or its subsidiaries in the United States and other countries.
- LMS Virtual.Lab is a registered trademark of LMS International N.V.
- MATLAB and Simulink are registered trademarks of The MathWorks, Inc.
- Microsoft, Windows, Windows2000, Windows XP, and VisualC++ are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries.
- PSIM is a registered trademark of Powersim Inc.
- UNIX is a registered trademark of The Open Group.
- All other product and company names are trademarks, registered trademarks, or service marks of their respective companies, organizations, or holders.
- This document does not include the letters ® and ™.

CONTENTS

Chapter 1. Introduction 1

1.1 About This Book 1

1.2 Notational Conventions 4

Chapter 2. Materials 5

2.1 JMAG-Designer Settings 6

2.1.1 [Material Editor] Dialog Box 6

2.1.2 Material Database 9

2.2 Magnetizing Properties of a Permanent Magnet 10

2.2.1 Important Notes Regarding Analysis for Using Permanent Magnet 10

2.2.2 Types of Defining Method for Magnetic Properties . . 10

2.2.3 Reversible and Irreversible Settings for a Permanent Magnet 15

2.2.4 Specifying the Linear Magnetizing Properties of a Permanent Magnet 16

2.2.5 Specifying the Nonlinear Magnetizing Properties of a Permanent Magnet 16

2.2.6 Correcting the B-H Curve (JMAG-Studio) 19

2.3 Magnetizing Properties of a Soft Magnetic Material . . . 20

2.3.1 Important Notes Regarding Analysis for Using Soft Magnetic Material 20

2.3.2 Types of Defining Method for Magnetizing Properties 21

2.3.3 Nonlinear Magnetizing Properties for a Soft Magnetic Material 25

2.3.4 Magnetizing Properties for an Anisotropic Magnetic Steel Sheet 27

2.3.5 Specifying the Magnetic Properties of a Stress Dependency 29

2.3.6 Specifying the Magnetic Properties of a Temperature Dependency 30

2.4 Electric Properties 31

2.4.1 Specifying Electrical Conductivity/Electric Resistivity (Magnetic Field Analysis) 31

2.4.2 Specifying Electric Conductivity/Electric Resistivity (Electric Field Analysis, Charge Distribution Analysis) 32

2.4.3 Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis) 33

2.4.4 Specifying Anisotropy for the Electric Properties (JMAG-Studio) 34
2.4.5 Specifying Relative Permittivity (Electric Field Analysis, Charge Distribution Analysis) 34
2.4.6 Specifying Permittivity Settings (Structural Analysis) 35
2.5 Thermal Properties 36
2.5.1 Specifying Thermal Conductivity 36
2.5.2 Specifying Specific Heat . . 37
2.6 Mechanical Properties 38
2.6.1 Specifying the Density (Magnetic Field Analysis) . 38
2.6.2 Specifying the Density (Thermal Analysis) 38
2.6.3 Specifying the Magnetostriction (Magnetic Field Analysis) . 38
2.6.4 Specifying Mechanical Properties (Isotropic/ Anisotropy) (Magnetic Field Analysis) 39
2.6.5 Specifying the Mechanical Properties (Isotropic) (Structural Analysis) 40
2.6.6 Specifying the Mechanical Properties (Anisotropy) (Structural Analysis) 40
2.6.7 Specifying the Piezoelectric Stress Constant (Structural Analysis). 42
2.7 Material Database 44
2.7.1 Function 44
2.7.2 Material Database Table . . 45
2.7.3 China Steel 47
2.7.4 DAIDO ELECTRONICS 47
2.7.5 Daido Steel 49
2.7.6 Hoganas AB 50
2.7.7 Hitachi Metals 51
2.7.8 JFE Ferrite 52
2.7.9 JFE Steel 53
2.7.10 MITSUBISHI STEEL 54
2.7.11 Nippon Steel 54
2.7.12 Shin-Etsu Chemical 55
2.7.13 Sumitomo Metal Industries 55
2.7.14 TDK 56
2.7.15 Tokyo Ferrite 56
2.7.16 VAC 57
2.7.17 JSOL (Sample Material Data) 58
2.8 Coordinate System 59
2.8.1 Rectangular Coordinate System 59
2.8.2 Cylindrical Coordinate System 60
2.9 Material Types of Magnetic Field Analysis (JMAG-Studio) 61
2.10 Superconductivity Properties (JMAG-Studio) 64
2.10.1 Superconductor Model Types 64
2.10.2 Critical Current Density . . . 66
2.10.3 Setting Anisotropy for the Electrical Conductivity of Superconductors 66
2.10.4 Notes on Subdividing the Elements 67

2.11 Gap Element (JMAG-Studio) 68

Chapter 3. Function of Magnetic Field Analysis 69

3.1 Overview of Functions 69

3.1.1 ST (Magnetic Field Analysis (Static Analysis) Module) . 70

3.1.2 TR/DP (Magnetic Field Analysis (Transient Response Analysis) Module) 70

3.1.3 FQ (Magnetic Field Analysis (Frequency Response Analysis) Module) 71

3.1.4 TS (Transformer Analysis Module) (JMAG-Designer) 72

3.2 Conditions of Magnetic Field Analysis (JMAG-Designer) . . 73

3.3 Conditions of Magnetic Field Analysis (JMAG-Studio) 76

3.4 Material Property for Each Part: Permanent Magnet . . . 78

3.4.1 Specifying the Magnetization Direction of a Permanent Magnet 78

3.4.2 Setting the Skew for the Magnetization Direction of a Permanent Magnet . . 88

3.4.3 Using the Magnetizing Tool for a Permanent Magnet (JMAG-Studio) 93

3.4.4 Correcting the Magnetizing Properties of a Permanent Magnet 93

3.4.5 Specifying Isotropy or Anisotropy of a Permanent Magnet 94

3.5 Material Property for Each Part: Soft Magnetic Material . 97

3.5.1 Calculating Magnetostrictive Force 97

3.5.2 Specifying Lamination Direction for Magnetic Steel 97

3.5.3 Correcting the Magnetizing Properties of a Soft Magnetic Material 98

3.6 Material Property for Each Part: Electric Field Analysis . . 99

3.6.1 Allowing Eddy Current . . . 99

3.6.2 Allowing Electric Insulation 99

3.6.3 Using SIBC (Surface Impedance Method) 100

3.6.4 Using Laminated Structures of a Magnetic Steel Sheet during Eddy Current Calculation (JMAG-Designer) 100

3.6.5 Allowing Displacement Current in the Frequency Response Analysis (JMAG-Studio) 101

3.7 Periodic Boundary (Rotation/ Translation) 102

3.7.1 Function 102

3.7.2 Important Notes Regarding the Use of This Function . 103

3.8 Symmetry Boundary/Natural Boundary 108

3.8.1 Function 108

3.8.2 Important Notes Regarding the Use of This Function . 109

3.9 Gap Flux Boundary (JMAG-Designer) 111

3.9.1 Function 111

3.9.2 Important Notes Regarding the Use of This Function . 111

3.10 Current 113

3.10.1 Function 113

3.10.2 Important Notes Regarding the Use of This Function . 115

3.11 Current Density 125

3.11.1 Function 125

3.11.2 Important Notes Regarding the Use of This Function . 126

3.12 FEM Coil 131

3.12.1 Function 131

3.12.2 Important Notes Regarding the Use of This Function . 132

3.13 Group (FEM Coil) 140

3.13.1 Function 140

3.13.2 Important Notes Regarding the Use of This Function . 140

3.14 FEM Conductor 141

3.14.1 Function 141

3.14.2 Important Notes Regarding the Use of This Function . 142

3.15 Group (FEM Conductor) . . . 146

3.15.1 Function 146

3.15.2 Important Notes Regarding the Use of This Function . 146

3.16 External Circuit 147

3.16.1 Function 147

3.16.2 Important Notes Regarding the Use of This Function . 149

3.16.3 FEM Coil Component 151

3.16.4 FEM Conductor Component 153

3.16.5 Reference Coil Component/ Reference Conductor Component 153

3.16.6 Resistor Component 156

3.16.7 Capacitor Component . . 159

3.16.8 Inductor Component 159

3.16.9 Switch Component 159

3.16.10 User Subroutine Component (JMAG-Studio) 162

3.16.11 Diode Component 162

3.16.12 Electric Potential Source (1 Terminal) Component . 165

3.16.13 Current Source (1 Terminal) Component 168

3.16.14 Electric Potential Source (2 terminals) Component 171

3.16.15 Power Source Component (JMAG-Designer) 172

3.16.16 Brush/Commutator Component 172

3.16.17 Three-phase Electric Potential Source Component / Three-phase Current Source Component 178

3.16.18 Macro Component 179

3.16.19 Electric Potential Probe Component 179

3.16.20 Current Probe Component 179

3.16.21 Ground Component 179

3.16.22 Circuit Component Types and Settings 180

3.17 Motion (Rotation/ Translation) 182

3.17.1 Function 182

3.17.2 Important Notes Regarding the Use of This Function . 184

3.17.3 Generating Mesh with the Motion Condition Set . . . 205

3.18 Slide 212

3.18.1 Function 212

3.18.2 Important Notes Regarding the Use of This Function . 213

3.19 Electromagnetic Force and Torque Calculation 216

3.19.1 Function 216

3.19.2 Important Notes Regarding the Use of This Function . 217

3.20 Heat Source Calculation . . . 224

3.20.1 Function 224

3.20.2 Important Notes Regarding the Use of This Function . 225

3.21 Iron Loss Calculation 226

3.21.1 Function 226

3.21.2 Important Notes Regarding the Use of This Function . 228

3.22 Magnetic Flux Calculation . 230

3.22.1 Function 230

3.22.2 Important Notes Regarding the Use of This Function . 230

3.23 Magnetostriction Calculation (JMAG-Designer) 232

3.23.1 Function 232

3.23.2 Important Notes Regarding This Function 233

3.24 Partial Model 235

3.24.1 Function 235

3.24.2 Important Notes Regarding the Use of This Function . 236

3.25 External Circuit Link (Simulink, PSIM) 238

3.25.1 Function 238

3.25.2 Important Notes Regarding the Use of This Function . 239

3.25.3 Linking to Simulink 240

3.25.4 Linking to PSIM 243

3.26 Subcycling 246

3.26.1 Function 246

3.26.2 Important Notes Regarding the Use of This Function . 247

3.26.3 Analysis Flow Using the Subcycling Condition 249

3.27 Frozen Permeability (JMAG-Designer) 251

3.27.1 Function 251

3.27.2 Important Notes Regarding the Use of This Function . 252

3.28 Stress Distribution 253

3.28.1 Function 253

3.28.2 Important Notes Regarding the Use of This Function . 254

3.29 Temperature Distribution (JMAG-Designer) 255

3.29.1 Function 255

3.29.2 Important Notes Regarding the Use of This Function . 255

3.30 Displacement (JMAG-Designer) 256

3.30.1 Function 256
3.30.2 Important Notes Regarding the Use of This Function . 257

3.31 Reusing Demagnetization (JMAG-Designer) 258

3.31.1 Function 258
3.31.2 Important Notes Regarding This Function 259

3.32 External Field 260

3.32.1 Function 260
3.32.2 Important Notes Regarding the Use of This Function . 261

3.33 Gap 262

3.33.1 Function 262
3.33.2 Important Notes Regarding the Use of This Function . 263

3.34 Insulation 264

3.34.1 Function 264
3.34.2 Important Notes Regarding the Use of This Function . 265

3.35 Partial Inductance 267

3.35.1 Function 267
3.35.2 Important Notes Regarding the Use of This Function . 268

3.36 Analysis Type 269

3.36.1 Function 269
3.36.2 Important Notes Regarding the Use of This Function . 269

3.37 Calculation Method 270

3.37.1 Function 270
3.37.2 Important Notes Regarding the Use of This Function . 271

3.38 Step Control 272

3.38.1 Function 272
3.38.2 Important Notes Regarding This Function 274

3.39 Frequency Control 278

3.39.1 Function 278
3.39.2 Important Notes Regarding This Function 279

3.40 Full Model Conversion 283

3.40.1 Function 283

3.41 Two-Way Coupled Analysis . 284

3.41.1 Function 284
3.41.2 Important Notes Regarding the Use of This Function . 285

3.42 Circuit Conversion 286

3.42.1 Function 286
3.42.2 Important Notes Regarding the Use of This Function . 287

3.43 Power Source Component Control (JMAG-Designer) . . 293

3.43.1 Function 293
3.43.2 Important Notes Regarding This Function 293

3.44 Parallel Computing: Shared Memory Multiprocessing (SMP) 294

3.44.1 Function 294

3.44.2 Important Notes Regarding the Use of This Function . 294

3.45 Parallel Computing: Distributed Memory Multiprocessor (DMP) 297

3.45.1 Function 297

3.45.2 Important Notes Regarding the Use of This Function . 297

3.46 Use GPU (JMAG-Designer) . 300

3.46.1 Function 300

3.46.2 Important Notes Regarding the Use of This Function . 300

3.47 Steady-state Approximate Transient Analysis 301

3.47.1 Function 301

3.47.2 Important Notes Regarding the Use of This Function . 302

3.48 Time Periodic Explicit Error Correction (JMAG-Designer) 303

3.48.1 Function 303

3.48.2 Important Notes Regarding the Use of This Function . 304

3.49 Displacement Current (JMAG-Studio) 305

3.49.1 Function 305

3.49.2 Important Notes Regarding the Use of This Function . 305

3.50 ICCG 306

3.50.1 Function 306

3.50.2 Important Notes Regarding the Use of This Function . 307

3.51 Nonlinear Calculation 311

3.51.1 Function 311

3.51.2 Important Notes Regarding the Use of This Function . 312

3.52 Post Output 316

3.52.1 Function 316

3.52.2 Important Notes Regarding the Use of This Function . 317

3.53 Controlling User Subroutine 323

3.53.1 Function 323

3.53.2 Important Notes Regarding the Use of This Function . 324

3.54 Snapshot Calculation 325

3.54.1 Function 325

3.54.2 Important Notes Regarding the Use of This Function . 325

3.54.3 Executing Analysis using Third-party Job Management Systems . . . 326

3.55 Restart 328

3.55.1 Function 328

3.55.2 Types of Restart 329

3.56 User Subroutine 330

3.56.1 Types of User Subroutine 330

3.56.2 Environment Required for the User Subroutine 333

3.56.3 Building User Subroutines Using C Language 333

3.56.4 Building User Subroutines Using Fortran 337

3.56.5 User Subroutine Data in the Report File 341

3.56.6 Building a User Subroutine (Linux) 343

3.56.7 magusr 345

3.56.8 conusr 349

3.56.9 usrfma 351
3.56.10 usrfm2 359
3.56.11 usrcir 364
3.56.12 usrstp 366
3.56.13 usrfle 367
3.56.14 usrxv 368
3.56.15 usrlos 369
3.56.16 usrmtx 371
3.56.17 hsusr1 373
3.56.18 hsusr3 375
3.56.19 hsusr4 377
3.56.20 usrhys 379
3.56.21 usrene 381
3.56.22 usrplt 383
3.56.23 usrsc (JMAG-Studio) 385
3.56.24 usrstr 386

3.57 DLL of Magnetic Field Analysis (jbdll.dll) 390

3.57.1 Function 390

3.58 Setting Environment Variables 408

3.58.1 Using a Batch File to Set Environmental Variables . 408
3.58.2 Setting the Environmental Variables on Your Computer (For Windows XP) 409
3.58.3 Checking the Registry (For Windows XP) 409

3.59 Output Items of Magnetic Field Analysis 410

3.59.1 View Type and Display Method 410
3.59.2 Output Items 411

Chapter 4. Function of Thermal Analysis 419

4.1 Overview of Functions 419

4.1.1 HT (Thermal Analysis Module) 419

4.2 Conditions of Thermal Analysis (JMAG-Designer) . . 420

4.3 Conditions of Thermal Analysis (JMAG-Studio) 422

4.4 Material Property for Each Part 423

4.4.1 Endotherm Coefficient and Equilibrium Temperature 423
4.4.2 Output Control (JMAG-Studio) 423

4.5 Periodic Boundary (Rotation/ Translation) 424

4.5.1 Function 424
4.5.2 Important Notes Regarding the Use of This Function . 425

4.6 Natural Boundary (JMAG-Designer) 427

4.6.1 Function 427

4.7 Constant Temperature Boundary 428

4.7.1 Function 428
4.7.2 Important Notes Regarding This Function 428

4.8 Equivalent Temperature Boundary 429

4.8.1 Function 429
4.8.2 Important Notes Regarding This Function 430

4.9 Heat Transfer Boundary . . . 431
4.9.1 Function 431
4.9.2 Important Notes Regarding the Use of This Function . 432
4.10 Radiation Heat Transfer Boundary 434
4.10.1 Function 434
4.10.2 Important Notes Regarding the Use of This Function . 435
4.11 Heat Flux Boundary 438
4.11.1 Function 438
4.11.2 Important Notes Regarding the Use of This Function . 438
4.12 Initial Temperature 439
4.12.1 Function 439
4.12.2 Important Notes Regarding the Use of This Function . 439
4.13 Heat Source (Heat Source Density/Heat Source Amount/Heat Source Density Coefficient) 441
4.13.1 Function 441
4.13.2 Important Notes Regarding the Use of This Function . 442
4.14 Thermal Circuit 444
4.14.1 Function 444
4.14.2 Important Notes Regarding the Use of This Function . 445
4.14.3 Circuit Components 445
4.15 Motion (Rotation/ Translation) 448
4.15.1 Function 448
4.15.2 Important Notes Regarding the Use of This Function . 449
4.16 Average Temperature Calculation 450
4.16.1 Function 450
4.16.2 Important Notes Regarding the Use of This Function . 450
4.17 Heat Flow Calculation 451
4.17.1 Function 451
4.17.2 Important Notes Regarding the Use of This Function . 451
4.18 Contact Thermal Resistance 452
4.18.1 Function 452
4.18.2 Important Notes Regarding the Use of This Function . 453
4.19 Partial Model 454
4.19.1 Function 454
4.19.2 Important Notes Regarding the Use of This Function . 455
4.20 Analysis Type 457
4.20.1 Function 457
4.20.2 Important Notes Regarding This Function 457
4.21 Step Control 458
4.21.1 Function 458
4.21.2 Important Notes Regarding the Use of This Function . 458
4.22 Two-Way Coupled Analysis .460
4.22.1 Function 460
4.23 ICCG 461
4.23.1 Function 461

4.23.2 Important Notes Regarding the Use of This Function . 461

4.24 Nonlinear Calculation 462

4.24.1 Function 462

4.24.2 Important Notes Regarding the Use of This Function 462

4.25 Output Control 464

4.25.1 Function 464

4.25.2 Important Notes Regarding the Use of This Function . 464

4.25.3 Controlling the File Size (JMAG-Studio) 465

4.26 Parallel Computing: Shared Memory Multiprocessing (SMP) 467

4.26.1 Function 467

4.26.2 Important Notes Regarding the Use of This Function . 467

4.27 Controlling User Subroutine 468

4.27.1 Function 468

4.27.2 Important Notes Regarding the Use of This Function . 468

4.28 Restart 470

4.28.1 Function 470

4.29 User Subroutine 471

4.29.1 Types of User Subroutine 471

4.29.2 dtuser 473

4.29.3 usrstp 474

4.29.4 usrhtc 475

4.29.5 usrhrd 476

4.29.6 ucond 478

4.29.7 uspeci 480

4.29.8 flgph 483

4.29.9 phase1 485

4.29.10 phase2 489

4.29.11 udnsty 493

4.29.12 quser 494

4.30 Output Items of Thermal Analysis 497

Chapter 5. Function of Electric Field Analysis 499

5.1 Overview of Functions 499

5.1.1 EL (Electric Field Analysis Module) 499

5.1.2 CH (Charge Distribution Analysis Module) 500

5.2 Conditions of Electric Field Analysis (JMAG-Designer) . . 501

5.3 Conditions of Electric Field Analysis (JMAG-Studio) 503

5.4 Material Property for Each Part 505

5.4.1 Specifying the Particle Flight Region (Charge Distribution Analysis) 505

5.4.2 Post Output (JMAG-Studio) 505

5.5 Periodic Boundary (Rotation/ Translation) 506

5.5.1 Function 506

5.5.2 Important Notes Regarding the Use of This Function . 507

5.6 Symmetry Boundary/Natural Boundary (JMAG-Designer) . 510

5.6.1 Function 510

5.6.2 Important Notes Regarding the Use of This Function . 510

5.7 Electric Potential Boundary (EL) 511

5.7.1 Function 511

5.7.2 Important Notes Regarding the Use of This Function . 512

5.8 Electric Potential Boundary (CH) 513

5.8.1 Function 513

5.8.2 Important Notes Regarding the Use of This Function . 513

5.9 Electric Field Boundary 514

5.9.1 Function 514

5.9.2 Important Notes Regarding the Use of This Function . 514

5.10 Current Boundary (JMAG-Designer) 515

5.10.1 Function 515

5.10.2 Important Notes Regarding the Use of This Function . 515

5.11 Current Density Boundary . 516

5.11.1 Function 516

5.11.2 Important Notes Regarding the Use of This Function . 516

5.12 Force Calculation by Electric Field 517

5.12.1 Function 517

5.12.2 Important Notes Regarding the Use of This Function . 517

5.13 Surface Charge Distribution Calculation (Surface Charge Density) 518

5.13.1 Function 518

5.13.2 Important Notes Regarding the Use of This Function . 518

5.14 Current Calculation 519

5.14.1 Function 519

5.14.2 Important Notes Regarding the Use of This Function . 519

5.15 Conductor 521

5.15.1 Function 521

5.15.2 Important Notes Regarding the Use of This Function . 521

5.16 Insulation 522

5.16.1 Function 522

5.16.2 Important Notes Regarding the Use of This Function . 522

5.17 Face Charge (Face Charge Density) 523

5.17.1 Function 523

5.17.2 Important Notes Regarding the Use of This Function . 523

5.18 Volume Charge (Charge Density) 524

5.18.1 Function 524

5.18.2 Important Notes Regarding the Use of This Function . 524

5.19 Temperature Distribution (JMAG-Designer) 525

5.19.1 Function 525

5.19.2 Important Notes Regarding the Use of This Function . 525

5.20 Control Data of Charge Distribution Analysis (Particle Data) (JMAG-Studio) 526

5.20.1 Function 526
5.20.2 Important Notes Regarding the Use of This Function . 527
5.21 Initial Particle Density (JMAG-Studio) 528
5.21.1 Function 528
5.21.2 Important Notes Regarding the Use of This Function . 528
5.22 Analysis Type 529
5.22.1 Function 529
5.22.2 Important Notes Regarding the Use of This Function . 529
5.23 Step Control / Frequency Control 530
5.23.1 Function 530
5.23.2 Important Notes Regarding the Use of This Function . 531
5.24 Two-Way Coupled Analysis 532
5.24.1 Function 532
5.25 ICCG 533
5.25.1 Function 533
5.25.2 Important Notes Regarding the Use of This Functions 534
5.26 Nonlinear Calculation 535
5.26.1 Function 535
5.26.2 Important Notes Regarding the Use of This Function . 535
5.27 Post Output 537
5.27.1 Function 537
5.27.2 Important Notes Regarding the Use of This Function . 538
5.28 Parallel Computing: Shared Memory Multiprocessing (SMP) 539
5.28.1 Function 539
5.28.2 Important Notes Regarding the Use of This Function . 539
5.29 User Subroutine (Current Distribution Analysis: EL) . . . 540
5.29.1 Types of User Subroutines 540
5.29.2 Required Environment for User Subroutine 540
5.29.3 Notes on Using User Subroutines 540
5.29.4 Reconfiguration of JMAG 541
5.29.5 cndusr 543
5.29.6 usesee 544
5.29.7 usrsee 545
5.30 User Subroutine (Charge Distribution Analysis: CH) (JMAG-Studio) 546
5.30.1 Types of User Subroutine 547
5.30.2 Notes on Using User Subroutines 549
5.30.3 Reconfiguration of JMAG-Studio 549
5.30.4 usrdns 552
5.30.5 velusr 554
5.30.6 rgtusr 556
5.30.7 dtuser 558
5.30.8 setdns 560
5.30.9 bmuser 562
5.30.10 usrgap 564
5.31 Output Items of Electric Field Analysis 569

Chapter 6. Function of Structural Analysis (JMAG-Designer) 573

6.1 Overview of Functions 573

6.1.1 DS (Structural Analysis Module) 573

6.2 Conditions of Structural Analysis (JMAG-Designer) . 575

6.3 Material Property for Each Part: Beam 577

6.3.1 Function 577

6.3.2 Important Notes Regarding the Use of This Function . 578

6.4 Material Property for Each Part: Shell 581

6.4.1 Function 581

6.4.2 Important Notes Regarding the Use of This Function . 582

6.5 Material Property for Each Part: Solid 583

6.5.1 Function 583

6.5.2 Important Notes Regarding the Use of This Function . 583

6.6 Electric Potential Boundary (Piezoelectric analysis) 584

6.6.1 Function 584

6.6.2 Important Notes Regarding This Function 585

6.7 Concentrated Load 586

6.7.1 Function 586

6.8 Pressure Load 587

6.8.1 Function 587

6.9 Volume Load 588

6.9.1 Function 588

6.9.2 Important Notes Regarding the Use of This Function . 588

6.10 Acceleration Load 589

6.10.1 Function 589

6.11 Centrifugal Force 590

6.11.1 Function 590

6.11.2 Important Notes Regarding the Use of This Function . 590

6.12 Electromagnetic Force 591

6.12.1 Function 591

6.12.2 Important Notes Regarding the Use of This Function . 592

6.13 Temperature Load 593

6.13.1 Function 593

6.13.2 Important Notes Regarding the Use of This Function . 594

6.14 Press Fit 595

6.14.1 Function 595

6.14.2 Important Notes Regarding the Use of This Function . 595

6.15 Displacement 596

6.15.1 Function 596

6.15.2 Important Notes Regarding the Use of This Function . 597

6.16 Rigid Body 599

6.16.1 Function 599

6.16.2 Important Notes Regarding the Use of This Function . 599

6.17 Constraint 600

6.17.1 Function 600
6.17.2 Important Notes Regarding the Use of This Function . 601
6.18 Periodic Boundary (Rotation) 602
6.18.1 Function 602
6.18.2 Important Notes Regarding the Use of This Function . 603
6.19 Adhesion 604
6.19.1 Function 604
6.19.2 Important Notes Regarding the Use of This Function . 605
6.20 Spring 606
6.20.1 Function 606
6.20.2 Important Notes Regarding the Use of This Function . 606
6.21 Concentrated Mass 607
6.21.1 Function 607
6.22 Sound Pressure 608
6.22.1 Function 608
6.22.2 Important Notes Regarding the Use of This Function . 609
6.23 Solver Calculation Control . 610
6.23.1 Function 610
6.23.2 Important Notes Regarding the Use of This Function . 611
6.24 Eigen Value 612
6.24.1 Function 612
6.24.2 Important Notes Regarding the Use of This Function . 612
6.25 Frequency Control 613
6.25.1 Function 613
6.25.2 Important Notes Regarding the Use of This Function . 614
6.26 Output Control 616
6.26.1 Function 616
6.27 Output Items of Structural Analysis 617

Chapter 7. Function of Structural Analysis (JMAG-Studio) 619
7.1 Overview of Functions 619
7.1.1 DS (Structural Analysis Module) 619
7.2 Conditions of Structural Analysis (JMAG-Studio) 620
7.3 Comment $ 623
7.3.1 Function 623
7.3.2 Important Notes Regarding the Use of This Function . 623
7.4 Solver Type SOL 624
7.4.1 Function 624
7.4.2 Important Notes Regarding the Use of This Function . 624
7.5 Parameters for Eigen Value Calculation $subite, $subtol 625
7.5.1 Function 625
7.5.2 Important Notes Regarding the Use of This Function . 625
7.6 End of Control Section CEND 626
7.6.1 Function 626

7.6.2 Important Notes Regarding the Use of This Function . 626

7.7 Frequency ID FREQ 627

7.7.1 Function 627

7.7.2 Important Notes Regarding the Use of This Function . 627

7.8 Eigen Value ID METH 628

7.8.1 Function 628

7.8.2 Important Notes Regarding the Use of This Function . 628

7.9 Damping Coefficient ID SDAM 629

7.9.1 Function 629

7.9.2 Important Notes Regarding the Use of This Function . 629

7.10 Dynamic Load ID DLOAD . 630

7.10.1 Function 630

7.10.2 Important Notes Regarding the Use of This Function . 630

7.11 Single Point Constraint ID SPC 631

7.11.1 Function 631

7.11.2 Important Notes Regarding the Use of This Function . 631

7.12 Force ID LOAD 632

7.12.1 Function 632

7.12.2 Important Notes Regarding the Use of This Function . 632

7.13 Temperature ID TEMPERATURE 633

7.13.1 Function 633

7.13.2 Important Notes Regarding the Use of This Function . 633

7.14 Multipoint Constraint ID MPC 634

7.14.1 Function 634

7.14.2 Important Notes Regarding the Use of This Function . 634

7.15 Controlling Output Items velo,acce,stre 635

7.15.1 Function 635

7.15.2 Important Notes Regarding the Use of This Function . 635

7.16 Frequency Range FREQ1 . .636

7.16.1 Function 636

7.17 Eigen Value EIGRL 637

7.17.1 Function 637

7.17.2 Important Notes Regarding the Use of This Function . 637

7.18 Modal Damping Table TABDMP1 638

7.18.1 Function 638

7.18.2 Important Notes Regarding the Use of This Function . 638

7.19 Solid Element Properties PSOLID 639

7.19.1 Function 639

7.19.2 Important Notes Regarding the Use of This Function . 639

7.20 Shell Element Properties PSHELL 640

7.20.1 Function 640

7.20.2 Important Notes Regarding the Use of This Function . 640

7.21 Beam Element Properties PBAR 641
7.21.1 Function 641
7.21.2 Important Notes Regarding the Use of This Function . 641
7.22 Isotropic Material MAT1 . 643
7.22.1 Function 643
7.22.2 Important Notes Regarding the Use of This Function . 644
7.23 Anisotropic Material 2D MAT2, JMAT2 645
7.23.1 Function 645
7.23.2 Important Notes Regarding the Use of This Function . 646
7.24 Anisotropic Material 3D MAT9, JMAT9 647
7.24.1 Function 647
7.24.2 Important Notes Regarding the Use of This Function . 649
7.25 Nodal Coordinates GRID . 650
7.25.1 Function 650
7.26 6-sided Solid Element CHEXA 651
7.26.1 Function 651
7.26.2 Important Notes Regarding the Use of This Function . 651
7.27 4-sided Solid Element CTETRA 652
7.27.1 Function 652
7.27.2 Important Notes Regarding the Use of This Function . 652
7.28 5-sided Solid Element CPENTA 653
7.28.1 Function 653
7.28.2 Important Notes Regarding the Use of This Function . 653
7.29 Quadrilateral Shell Element CQUAD4 654
7.29.1 Function 654
7.29.2 Important Notes Regarding the Use of This Function . 654
7.30 Triangular Shell Element CTRIA3 655
7.30.1 Function 655
7.30.2 Important Notes Regarding the Use of This Function . 655
7.31 Beam Element CBAR 656
7.31.1 Function 656
7.31.2 Important Notes Regarding the Use of This Function . 656
7.32 Dynamic Load ID DLOAD . 657
7.32.1 Function 657
7.32.2 Important Notes Regarding the Use of This Function . 657
7.33 Dynamic Load RLOAD1 . . .658
7.33.1 Function 658
7.33.2 Important Notes Regarding the Use of This Function . 658
7.34 Dynamic Load: Scaling Factor DAREA 659
7.34.1 Function 659
7.34.2 Important Notes Regarding the Use of This Function . 659
7.35 Dynamic Load: Phase Angle DPHASE 660

7.35.1 Function 660
7.35.2 Important Notes Regarding the Use of This Function . 660

7.36 Dynamic Load: Table TABLED1 661

7.36.1 Function 661
7.36.2 Important Notes Regarding the Use of This Function . 661

7.37 Force FORCE 662

7.37.1 Function 662
7.37.2 Important Notes Regarding the Use of This Function . 662

7.38 Centrifugal Force RFORCE 663

7.38.1 Function 663
7.38.2 Important Notes Regarding the Use of This Function . 663

7.39 Nodal Temperature TEMP 664

7.39.1 Function 664
7.39.2 Important Notes Regarding the Use of This Function . 664

7.40 Single Point Constraint SPC 665

7.40.1 Function 665
7.40.2 Important Notes Regarding the Use of This Function . 666

7.41 Rectangular Coordinate System CORD2R 668

7.41.1 Function 668

7.42 Cylindrical Coordinate System CORD2C 669

7.42.1 Function 669
7.42.2 Important Notes Regarding the Use of This Function . 669

7.43 Displacement Coordinate System NGRP 670

7.43.1 Function 670
7.43.2 Important Notes Regarding the Use of This Function . 671

7.44 Multipoint Constraint MPC 672

7.44.1 Function 672
7.44.2 Important Notes Regarding the Use of This Function . 673

7.45 Spring Element GSPRING .674

7.45.1 Function 674
7.45.2 Important Notes Regarding the Use of This Function . 675

7.46 Press Fit PRESS 676

7.46.1 Function 676
7.46.2 Important Notes Regarding the Use of This Function . 676

7.47 Concentrated Mass CONM2 677

7.47.1 Function 677

7.48 Rigid Body Element RBE2 .678

7.48.1 Function 678
7.48.2 Important Notes Regarding the Use of This Function . 679

7.49 Adhesion ASPRING 680

7.49.1 Function 680
7.49.2 Important Notes Regarding the Use of This Function . 681

7.50 Acoustic Analysis ACOUST 682

7.50.1 Function 682
7.50.2 Important Notes Regarding the Use of This Function . 682
7.51 Electromagnetic Force JLOAD 683
7.51.1 Function 683
7.51.2 Important Notes Regarding the Use of This Function . 685
7.52 Piezoelectric Constant MATP 686
7.52.1 Function 686
7.52.2 Important Notes Regarding the Use of This Function . 687
7.53 Relative Permittivity MATE 688
7.53.1 Function 688
7.53.2 Important Notes Regarding the Use of This Function . 689
7.54 Electrode ELCRD 690
7.54.1 Function 690
7.54.2 Important Notes Regarding the Use of This Function . 691
7.55 Pressure Load PLOAD . . . 692
7.55.1 Function 692
7.55.2 Important Notes Regarding the Use of This Function 693
7.56 Edge Force LINE_FORCE . . 694
7.56.1 Function 694
7.57 Volume Force VOLUME_FORCE 695
7.57.1 Function 695
7.57.2 Important Notes Regarding the Use of This Function . 695
7.58 Acceleration Force GRAV . 696
7.58.1 Function 696
7.59 End ENDDATA 697
7.59.1 Function 697
7.59.2 Important Notes Regarding the Use of This Function . 697
7.60 Output Items of Structural Analysis 698

Chapter 8. Function of Electromagnetic Wave Frequency Response Analysis (JMAG-Studio) 701
8.1 Overview of Functions 701
8.1.1 WV (Electromagnetic Wave Frequency Response Analysis Module) 701
8.2 Conditions of Electromagnetic Wave Frequency Response Analysis 702
8.3 Material Properties 703
8.3.1 Function 703
8.3.2 Important Notes Regarding the Use of This Function . 703
8.4 Perfect Electric Boundary (Symmetry Boundary) 705
8.4.1 Function 705
8.4.2 Important Notes Regarding the Use of This Function . 705
8.5 Perfect Magnetic Boundary (Natural Boundary) 706

8.5.1 Function 706
8.5.2 Important Notes Regarding the Use of This Function . 706

8.6 Transparent Boundary 707

8.6.1 Function 707
8.6.2 Important Notes Regarding the Use of This Function . 707

8.7 Surface Impedance Boundary 708

8.7.1 Function 708
8.7.2 Important Notes Regarding the Use of This Function . 708

8.8 PML Boundary (Perfect Matching Layer Boundary) . 709

8.8.1 Function 709
8.8.2 Important Notes Regarding the Use of This Function . 709

8.9 Periodic Boundary 711

8.9.1 Function 711
8.9.2 Important Notes Regarding the Use of This Function . 711

8.10 In/Out Port 714

8.10.1 Function 714
8.10.2 Important Notes Regarding the Use of This Function . 714

8.11 External Field 719

8.11.1 Function 719
8.11.2 Important Notes Regarding the Use of This Function . 719

8.12 External Field (Line Current) 720

8.12.1 Function 720
8.12.2 Important Notes Regarding the Use of This Function . 720

8.13 Voltage/Current Excitation .721

8.13.1 Function 721
8.13.2 Important Notes Regarding the Use of This Function . 721

8.14 Power Calculation 722

8.14.1 Function 722
8.14.2 Important Notes Regarding the Use of This Function . 722

8.15 Current Calculation 723

8.15.1 Function 723
8.15.2 Important Notes Regarding the Use of This Function . 723

8.16 Voltage Calculation 724

8.16.1 Function 724
8.16.2 Important Notes Regarding the Use of This Function . 724

8.17 Lumped Constant Component 725

8.17.1 Function 725
8.17.2 Important Notes Regarding the Use of This Function . 725

8.18 Laser Beam 726

8.18.1 Function 726
8.18.2 Important Notes Regarding the Use of This Function . 727

8.19 Plasma 728

8.19.1 Function 728
8.19.2 Important Notes Regarding the Use of This Function . 729

8.20 Frequency (Step) 731

8.20.1 Function 731
8.20.2 Important Notes Regarding the Use of This Function . 732
8.21 Analysis Control 733
8.21.1 Function 733
8.21.2 Important Notes Regarding the Use of This Function . 734
8.22 Analysis Control: Control Output Items 739
8.22.1 Function 739
8.23 User Subroutine 740
8.23.1 Types of User Subroutine 740
8.23.2 Notes on Using User Subroutines 740
8.23.3 Using User Subroutines . . 740
8.23.4 Creating the User Subroutine DLL 741
8.23.5 incusr+XX 743
8.23.6 extusr 745
8.23.7 plsusr_ini 747
8.23.8 plsusr 749
8.23.9 epsusr 753
8.24 Far Field Calculation 756
8.24.1 Function 756
8.24.2 Important Notes Regarding the Use of This Function . 757
8.25 Equivalent Current Calculation 758
8.25.1 Function 758
8.25.2 Important Notes Regarding the Use of This Function . 758
8.26 Output Items of Electromagnetic Wave Frequency Response Analysis 759
Chapter 9. Mesh 761
9.1 Adaptive Mesh Control (Magnetic Field Analysis) . . 762
9.1.1 Function 762
9.1.2 Important Notes Regarding the Use of This Function . 763
9.2 Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (JMAG-Designer) . 767
9.2.1 Function 767
9.2.2 Important Notes Regarding This Function 767
Chapter 10. Coupled Analysis. 769
10.1 Types of Coupled Analysis . . 769
10.2 Solver Combinations for Coupled Analysis 771
10.3 Time Interval Settings for Two-way Coupled Analysis (JMAG-Designer) 773
10.4 Coupling Magnetic Field Analysis and Thermal Analysis in One Direction 778
10.4.1 Magnetic Field Analysis, Iron Loss Calculation => Thermal Analysis 779
10.4.2 Thermal Analysis => Magnetic Field Analysis (JMAG-Designer) 781

10.5 Coupling Magnetic Field Analysis and Thermal Analysis in Both Directions 783

10.5.1 Magnetic Field Analysis (Transient) (Loss Averaging Flag) <=> Thermal Analysis 783

10.5.2 Magnetic Field Analysis (Transient) <=> Thermal Analysis 791

10.5.3 Magnetic Field Analysis (Frequency) <=> Thermal Analysis 795

10.6 Coupling Magnetic Field Analysis and Structural Analysis in One Direction . . 801

10.6.1 Magnetic Field Analysis (Static) => Structural Analysis (Static), Magnetic Field Analysis (Transient) => Structural Analysis (Static/Frequency) 802

10.6.2 Magnetic Field Analysis (Frequency) -> Structural Analysis (Static/ Frequency) 807

10.6.3 Structural Analysis => Magnetic Filed Analysis, Iron Loss Analysis 810

10.7 Coupling Thermal Analysis and Electric Field Analysis in One Direction 813

10.7.1 Current Distribution Analysis, Electric Field Analysis (Frequency) => Thermal Analysis 813

10.7.2 Thermal Analysis => Current Distribution Analysis 815

10.8 Coupling Thermal Analysis and Electric Field Analysis in Both Directions 817

10.8.1 Thermal Analysis <=> Current Distribution Analysis, Electric Field Analysis (Frequency) 817

10.9 Coupling Thermal Analysis and Structural Analysis in One Direction 821

10.9.1 Thermal Analysis => Structural Analysis (JMAG-Designer) 821

Chapter 11. Coupled Analysis Using Abaqus 823

11.1 Types of Coupled Analysis . .823

11.2 One-way Coupled Analysis (JMAG > Abaqus) 824

11.2.1 JMAG Magnetic Field Analysis (Force) > Abaqus Structural Analysis 825

11.2.2 JMAG Magnetic Field Analysis, Iron Loss Analysis > Abaqus Heat Transfer Analysis 831

11.3 One-way Coupled Analysis (Abaqus > JMAG) 836

11.3.1 Abaqus Structural Analysis (stress) > JMAG Magnetic Field Analysis, Iron Loss Analysis 836

11.4 Two-way Coupled Analysis (JMAG < > Abaqus) 839

11.4.1 JMAG Magnetic Field Analysis (Joule Loss Density) < > Abaqus Fully Coupled Analysis for Thermal and Stress (Displacement/ Temperature) 840
11.4.2 JMAG Magnetic Field Analysis (Force) < > Abaqus Dynamic Analysis (Displacement) 847

Chapter 12. Tools 853

12.1 Listing of Tools 853

12.2 Magnetization for Magnetic Field Analysis 858

12.2.1 Function 858
12.2.2 Procedures for Executing the Tool (JMAG-Studio) . . 859
12.2.3 Important Notes Regarding the Use of This Function (JMAG-Studio) 863
12.2.4 User Subroutine setusr . . 868

12.3 Calculate Iron Loss 872

12.3.1 Function 872
12.3.2 Flow of Calculation (JMAG-Studio) 878
12.3.3 Files Used for Iron Loss Calculation (JMAG-Studio) 879
12.3.4 Starting/Exiting the Calculate Iron Loss Tool (JMAG-Studio) 887
12.3.5 Settings in the [Basic] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio) 890
12.3.6 Settings in the [Material] Tab in the [Calculate Iron Loss] Dialog Box (JMAG-Studio) 892
12.3.7 Settings in the [Calculation] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio) 898
12.3.8 Settings in the [Option] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio) 900
12.3.9 Settings in the [User Subroutine] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio) 902
12.3.10 Running Iron Loss Calculation (JMAG-Studio) 903
12.3.11 Displaying Results (Output Items of Iron Loss Analysis) 904
12.3.12 Equations for Iron Loss Calculations 906
12.3.13 Procedure for Evaluating the Loss 923
12.3.14 User Subroutine iron_str . 927

12.4 Calculate Particle Trajectory (JMAG-Studio) 933

12.4.1 Function 933
12.4.2 Important Notes Regarding the Use of This Function . 933
12.4.3 Procedures for Executing the Tool 934
12.4.4 Output Items 936

12.5 Calculate Field Value (JMAG-Studio) 937

12.5.1 Function 937

12.5.2 Important Notes Regarding the Use of This Function . 937
12.5.3 Procedures for Executing the Tool 939
12.5.4 Output Items 942
12.5.5 Output Files 943
12.5.6 Analysis Control File Format 944

12.6 Calculate Far Field (JMAG-Studio) 947

12.6.1 Function 947
12.6.2 Important Notes Regarding the Use of This Function . 947
12.6.3 Procedures for Executing the Tool 947
12.6.4 Output Items 948

12.7 Creating Averaged Loss (Power Loss File) (JMAG-Studio) 949

12.7.1 Function 949
12.7.2 Important Notes Regarding the Use of This Function . 949
12.7.3 Procedures 949
12.7.4 Output File 951

12.8 Temperature for Structural Analysis (JMAG-Studio) . . . 952

12.8.1 Function 952
12.8.2 Procedures 952
12.8.3 Output Items 953

12.9 Force Mapping for Structural Analysis 954

12.9.1 Function 954
12.9.2 Important Notes Regarding the Use of This Function . 955
12.9.3 Procedures for JMAG-Designer 956
12.9.4 Procedures for JMAG-Studio 959
12.9.5 Output File 961
12.9.6 Executing the Tool from the Command Line 962

12.10 Force Export for Virtual.Lab (JMAG-Designer) 966

12.10.1 Data Transferred Between Programs and where it is Received 966
12.10.2 Flow of Analysis Linked with Virtual.Lab 966
12.10.3 Important Notes Regarding the Force Export for Virtual.Lab 967
12.10.4 Running procedures for the Force Export for Virtual.Lab 967
12.10.5 Export Files 968

12.11 Exporting Multi-purpose Files (JMAG-Designer) 969

12.11.1 Function 969
12.11.2 Data that can be Exported 969
12.11.3 Important Notes Regarding the Use of This Function . 969
12.11.4 Running Procedures 970
12.11.5 Export Files 975
12.11.6 Running from the Command Line 976

12.12 Calculate Motor Inductance (For Three-phase Permanent Magnet Synchronous Motor) 977

12.12.1 Function 977

12.12.2 Flow of Calculation 978
12.12.3 Files Required to Use the Tool 978
12.12.4 Important Notes Regarding the Use of This Function . 978
12.12.5 Procedures 979
12.12.6 Output Items 981
12.12.7 Output Files 981
12.12.8 Executing the Tool from the Command Line 982
12.12.9 Executing Analysis using Third-party Job Management Systems . . . 985

12.13 Calculate Coil Inductance . . 987

12.13.1 Function 987
12.13.2 Flow of Calculation 987
12.13.3 Files Required to Use the Tool 987
12.13.4 Notes on the Usage 988
12.13.5 Procedures 988
12.13.6 Output Items 991
12.13.7 Output Files 992
12.13.8 Executing the Tool from the Command Line 993
12.13.9 Executing Analysis using Third-party Job Management Systems . . . 996

12.14 Calculate Busbar Inductance 998

12.14.1 Function 998
12.14.2 Flow of Calculation 998
12.14.3 Files Required to Use the Tool 999
12.14.4 Procedures for Executing the Tool 1000
12.14.5 Output Items 1003
12.14.6 Output Files 1004
12.14.7 Executing from the Command Line 1005
12.14.8 Executing Analysis using Third-party Job Management Systems 1008

12.15 Calculate Design Approximate (JMAG-Studio) 1010

12.15.1 Function 1010
12.15.2 Files Required to Use the Tool 1011
12.15.3 Procedures for Executing the Tool 1012

Chapter 13. JMAG-RT 1015

13.1 About JMAG-RT 1015

13.2 Models Available in JMAG-RT 1017

13.3 Programs Required for JMAG-RT 1018

13.3.1 Create a JMAG-RT Model (running the FEM Analysis with JMAG-RT) 1018
13.3.2 Run the Circuit/Control Simulation using a JMAG-RT Model 1018

13.4 Using JMAG-RT (JMAG-RT Library Manager) 1021

13.5 Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy) . . 1028

13.5.1 Important Notes Regarding the Use of This Function 1028

13.5.2 Procedures for Executing this Function 1028

13.6 Creating JMAG-RT Model from a CSV File 1033

13.6.1 Important Notes Regarding the Use of This Function 1033

13.6.2 Input File Format 1033

13.7 Updating JMAG-RT Model Data File Created by Previous Version 1034

13.7.1 Important Notes Regarding the Use of This Function 1034

13.7.2 Operation for JMAG-RT Data File Conversion . . . 1034

13.8 Three-phase PM Synchronous Motor 1036

13.8.1 Creating an FEM Model with JMAG 1036

13.8.2 Settings in the JMAG-RT Library Manager 1038

13.8.3 Circuit/Control Simulation with Simulink 1054

13.8.4 Circuit/Control Simulation with PSIM 1067

13.8.5 Circuit/Control Simulation Using Programs in C . . . 1072

13.9 Two-phase Stepping Motor 1073

13.9.1 Creating an FEM Model with JMAG 1073

13.9.2 Settings in the JMAG-RT Library Manager 1075

13.9.3 Circuit/Control Simulation with Simulink 1081

13.9.4 Circuit/Control Simulation with PSIM 1085

13.9.5 Circuit/Control Simulation Using Programs in C . . . 1089

13.10 Linear Solenoid 1090

13.10.1 Creating an FEM Model with JMAG 1090

13.10.2 Settings in JMAG-RT Library Manager 1091

13.10.3 Circuit/Control Simulation with Simulink 1097

13.10.4 Circuit/Control Simulation with PSIM 1100

13.10.5 Circuit/Control Simulation Using Programs in C . . . 1103

13.11 Linear Synchronous Motor 1104

13.11.1 Creating an FEM Model with JMAG 1104

13.11.2 Settings in the JMAG-RT Library Manager 1105

13.11.3 Circuit/Control Simulation with Simulink 1115

13.11.4 Circuit/Control Simulator with PSIM 1121

13.11.5 Circuit/Control Simulation Using Programs in C . . . 1125

13.12 SR Motor 1126

13.12.1 Notes on Using the SR Motor Model Type 1126

13.12.2 Creating an FEM Model with JMAG 1126

13.12.3 Settings in the JMAG-RT Library Manager 1128

13.12.4 Circuit/Control Simulation with Simulink 1133

13.13 Induction Motor 1139

13.13.1 Notes on Using the Induction Motor Model Type 1139

13.13.2 Creating an FEM Model with JMAG 1140

13.13.3 Settings in the JMAG-RT Library Manager 1141

13.13.4 Circuit/Control Simulation with Simulink 1144

13.14 Creating a Model for JMAG-RT Using the Command Line 1149

13.14.1 Defining a Point Sequence (PointArray) 1153

13.14.2 Parameters for the Three-phase PM Synchronous Motor (PMSMotor) 1154

13.14.3 Parameters for the Stepping Motor (SteppingMotor) . 1159

13.14.4 Parameters for Linear Solenoid (LinearSolenoid) 1161

13.14.5 Parameters for the Linear Synchronous Motor (LSMotor) 1163

13.14.6 Parameters for the SR Motor (SRMotor) 1166

13.14.7 Parameters for the Induction Motor (InductionMotor) 1168

13.15 Running Simulations Using Programs in C 1170

13.15.1 Flow of Simulations Using Programs in C 1170

13.15.2 Simulation of Programs in C Using JMAGRT_DLL.dll 1172

13.15.3 Functions in JMAGRT_DLL.dll 1173

13.15.4 Samples of Programs in C 1182

13.15.5 Building a Program in C . 1182

13.15.6 Debugging a Program in C 1183

13.16 Executing Analysis using Third-party Job Management Systems 1184

Chapter 14. JMAG-RT Viewer. 1187

14.1 About JMAG-RT Viewer . . . 1187

14.2 License for JMAG-RT Viewer 1189

14.3 Notes Regarding the JMAG-RT Model Data File 1190

14.4 Starting JMAG-RT Viewer . 1191

14.5 Screen Configuration 1192

14.5.1 Main Window of JMAG-RT Viewer 1192

14.5.2 Detailed Settings for the Option Window 1193

14.5.3 Map View Window 1194

14.6 Menu Bar 1195

14.6.1 Menu Bar of the Main Window 1195

14.6.2 Menu Bar of the Map View Window 1196

14.7 Running Procedures 1198

14.7.1 Importing a JMAG-RT Model Data File 1198

14.7.2 Displaying the Ld/Lq Map 1198
14.7.3 Specifying a Parameters for Torque-Speed Curve / Efficiency Map 1199
14.7.4 Saving the Settings for Parameters for Torque-Speed Curve / Efficiency Map to a File . 1202
14.7.5 Importing the Setting File of Parameters for Torque-Speed Curve / Efficiency Map 1202
14.7.6 Displaying a Torque-Speed Curve / Efficiency Map . . 1203
14.7.7 Displaying the Type of the Torque-Speed Curve / Efficiency Map 1204
14.7.8 Exporting a Report of Torque-Speed Curve/ Efficiency Map 1205
14.7.9 Exporting the Tabular Data of Torque-Speed Curve / Efficiency Map 1206
14.8 Motor Control Information 1207
14.8.1 Algorithm for the Calculation of the Torque-Speed Curve . . . 1207
14.8.2 Algorithm for the Calculation of the Efficiency Map 1208
14.8.3 Important Notes Regarding Motor Control and Limitations of Motor Control 1208
14.9 Executing from the Command Line 1213
14.9.1 Setting the License Check (License Waiting Function) 1216
14.10 Error / Warning Codes 1217
Chapter 15. Editing a JCF File (Simple FHI) 1221
15.1 Overview 1221
15.2 Environment Required for Editing JCF Files 1223
15.2.1 Programs 1223
15.2.2 Files 1223
15.3 Functions of JCF File Access Library (SimpleFHI) 1226
15.4 Using the JCF File Access Library (SimpleFHI) 1247
15.4.1 Using C Language 1247
15.4.2 Using Excel VBA 1252
Chapter 16. Extracting Data from a Result File (JMAG-Designer) 1255
16.1 Function 1255
16.2 Important Notes Regarding the Use of This Function . .1256
16.3 Running Procedures (Windows/Linux) 1256
16.3.1 Important Notes Regarding the Descriptions of Commands 1256
16.3.2 Extracting Result Data . . 1257
16.3.3 Obtaining List of Output Items 1261

16.3.4 Obtaining a List for a Model or a Study Included in the Project 1261
16.3.5 Obtaining Values Set in Each Parameter of a Project 1262

INDEX 1263

Chapter 1. Introduction

1-1. About This Book

JMAG is an electromagnetic analysis software package that supports users in developing and designing electronic and electric equipment such as motors, actuators, circuit components, and antennae. JMAG performs the following analytic functions.

- Magnetic field analysis
- Thermal analysis
- Electric field analysis
- Electromagnetic wave frequency response analysis *JMAG-Studio Only
- Structural analysis

You can also perform the coupled analyses listed below by combining the above functions.

- Magnetic field and thermal analysis
- Magnetic field and structural analysis
- Thermal and electric field analysis
- Thermal and electromagnetic wave frequency response analysis *JMAG-Studio Only
- Thermal and structural analysis

This manual, “JMAG User's Manual Solver”, describes in detail the materials and conditions for each type of analysis available with JMAG.

▶▶ Reference

- For coupled analysis, see "Chapter 10. Coupled Analysis" on page 769.
- For information about basic JMAG-Studio functions, shape/mesh creation, condition specification, and script usage, see the JMAG-Studio User's Manual.
- For information about basic JMAG-Designer functions, shape/mesh creation, condition specification, and script usage, see the JMAG-Designer User's Manual.

The contents of each chapter are as follows:

■ **Chapter 1 Introduction**

Describes how to use this book.

■ **Chapter 2 Materials**

Describes the settings of material properties.

■ **Chapter 3 Function of Magnetic Field Analysis**

Describes the conditions of magnetic field analysis.

■ **Chapter 4 Function of Thermal Analysis**

Describes the conditions of thermal analysis.

■ **Chapter 5 Function of Electric Field Analysis**

Describes the conditions of electric field analysis.

■ **Chapter 6 Function of Structural Analysis (JMAG-Designer)**

Describes the conditions of structural analysis in JMAG-Designer.

■ **Chapter 7 Function of Structural Analysis (JMAG-Studio)**

Describes the material properties and conditions of structural analysis in JMAG-Studio.

■ **Chapter 8 Function of Electromagnetic Wave Frequency Response Analysis (JMAG-Studio)**

Describes the material properties and conditions of electromagnetic frequency response analysis in JMAG-Studio.

■ **Chapter 9 Mesh**

Describes the functions of mesh generation.

■ Chapter 10 Coupled Analysis

Describes the types of coupled analysis that can be run in JMAG.

■ Chapter 11 Coupled Analysis Using Abaqus

Describes how to use a coupled analysis in Abaqus.

■ Chapter 12 Tools

Describes the type of each tool such as [Magnetization for Magnetic Field Analysis] and [Calculate Iron Loss].

■ Chapter 13 JMAG-RT

Describes the tools that enables circuit / control simulation from an external program such as Simulink or PSIM, when referring to results data of a FEM analysis using JMAG-Solver.

■ Chapter 14 JMAG-RT Viewer

Describes the tools to create a torque-speed curve or efficiency map by using a JMAG-RT model data file that is exported from JMAG-RT or JMAG-Express.

■ Chapter 15 Editing a JCF File (Simple FHI)

Describes how to edit a JCF file by using a file access library specifically developed for JCF files.

■ Chapter 16 Extracting Data from a Result File (JMAG-Designer)

Describes the tools to extract results data from a project file or a result file of JMAG-Designer.

Important **The contents of this User's Manual may be revised without prior notice in order to improve the functionality of the main body.**

1-2. Notational Conventions

This book uses the following standards.

■ Terms Used in "JMAG User's Manual"

The name of each dialog box and menu item from the menu bar is enclosed by a pair of square brackets.

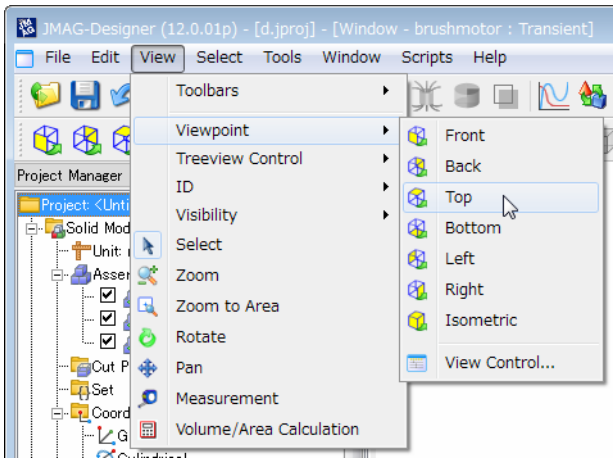
Examples: [Open] dialog box, [OK] button, [Basic] pane, and [Target] group box.

- In some sections, the control name is omitted as follows.
  Examples: Click [OK]. Click [Setting] in [Target].
- Operations other than JMAG are also enclosed by a pair of square brackets.

■ Operation from the Menu Bar

An operation selecting an objective item from the menu bar or a shortcut menu follows a hierarchical path of menu items down to reach an item required to perform an objective operation. This manual expresses a step of tracking down the hierarchical path by using a ">".

Examples: Select [View] > [Viewpoint] > [Top] from the menu bar.



■ Operation from the Keyboard

In JMAG, when using the extended function of each operation, you may need to press a key or keys on the keyboard. These function keys are presented in this manual enclosed by a pair of square brackets.

Examples: The [Ctrl] key, the [F1] key

Chapter 2. Materials

This chapter describes the material properties.

- Magnetizing Properties of a Permanent Magnet (10 page)
- Magnetizing Properties of a Soft Magnetic Material (20 page)
- Electric Properties (31 page)
- Thermal Properties (36 page)
- Mechanical Properties (38 page)
- Material Database (44 page)
- Coordinate System (59 page)
- Superconductivity Properties (64 page) * JMAG-Studio Only

▶▶ Reference

- For more information, see Chapter 27, “Creating a Material (Using the Material Database)” and Chapter 28, “Assigning Materials and Setting Material Properties” in the JMAG-Designer User's Manual.
- For more information about electromagnetic wave frequency response analysis, P.701 "Chapter 8. Function of Electromagnetic Wave Frequency Response Analysis (JMAG-Studio)".
- For more information about material properties of structural analysis for JMAG-Studio, see P.573 "Chapter 6. Function of Structural Analysis (JMAG-Designer)".

2-1. JMAG-Designer Settings

2-1-1. [Material Editor] Dialog Box

The available settings for each analysis in the [Material Editor] dialog are indicated in the following table.

Parameters	Modules *1								
	ST	TR/DP	FQ	LS	HT	EL-ST	EL-FQ	EL-CUR	DS
Magnetic Properties (Soft Magnetic Material)									
Anisotropy Type: Isotropic	o	o	o	o	x	x	x	x	x
Magnetic Property Type: Linear Constant	o	o	o	o	x	x	x	x	x
Magnetic Property Type: Linear (Temperature Dependent)	o	o	o	o	x	x	x	x	x
Magnetic Property Type: B-H Curve	o	o	o	o	x	x	x	x	x
Magnetic Property Type: B-M Curve	o	o	o	o	x	x	x	x	x
Magnetic Property Type: B-μ Curve	o	o	o	o	x	x	x	x	x
Magnetic Property Type: B-H Curve (Stress Dependent)	o	o	o	o	x	x	x	x	x
Magnetic Property Type: B-H Curve (Temperature Dependent)	o	o	o	o	x	x	x	x	x
Magnetic Property Type: User Subroutine magusr, usrmtx	o	o	o	o	x	x	x	x	x
Anisotropy Type: Anisotropic	o	o	o	o	x	x	x	x	x
Magnetic Properties (Permanent Magnet)									
Magnetic Property Type: Linear	o	o	x	x	x	x	x	x	x
Magnetic Property Type: Nonlinear (Reversible)	o	o	x	x	x	x	x	x	x
Magnetic Property Type: Nonlinear (Reversible/Temperature Dependent)	o	o	x	x	x	x	x	x	x
Magnetic Property Type: Irreversible Thermal Demagnetization with Demagnetizing Field	o	o	x	x	x	x	x	x	x
Magnetic Property Type: Irreversible Demagnetization with Demagnetizing Field	o	o	x	x	x	x	x	x	x
Magnetic Property Type: User Subroutine magusr, usrmtx	o	o	x	x	x	x	x	x	x

Parameters	Modules *1								
	ST	TR/DP	FQ	LS	HT	EL-ST	EL-FQ	EL-CUR	DS
Magnetic Properties (Magnetization)									
Initial Magnetization: Linear	o	o	x	x	x	x	x	x	x
Initial Magnetization: Nonlinear	o	o	x	x	x	x	x	x	x
Magnetization Ratio	o	o	x	x	x	x	x	x	x
Demagnetization: Linear	o	o	x	x	x	x	x	x	x
Demagnetization: Nonlinear	o	o	x	x	x	x	x	x	x
Electric Properties									
Electric Conductivity/Resistivity: Constant	o	o	o	x	x	x	o	o	x
Electric Conductivity/Resistivity: Temperature Dependent	o	o	o	x	x	x	o	o	x
Electric Conductivity/Resistivity: Electric Dependent	x	x	x	x	x	x	o	o	x
Electric Conductivity/Resistivity: Frequency Dependent	x	x	x	x	x	x	o	o	x
Electric Conductivity/Resistivity: User Subroutine conusr	o	o	o	x	x	x	x	x	x
Electric Conductivity/Resistivity: User Subroutine cndusr	x	x	x	x	x	x	o	x	x
Permittivity: Constant	x	x	x	x	x	o	o	x	o
Permittivity: Electric Dependent	x	x	x	x	x	o	o	x	x
Permittivity: Frequency Dependent	x	x	x	x	x	o	o	x	x
Permittivity: Component Input	x	x	x	x	x	x	x	x	o
Thermal Properties									
Thermal Conductivity: Constant	x	x	x	x	o	x	x	x	x
Thermal Conductivity: Temperature Dependent	x	x	x	x	o	x	x	x	x
Thermal Conductivity: User Subroutine ucond	x	x	x	x	o	x	x	x	x
Specific Heat: Constant	x	x	x	x	o	x	x	x	x
Specific Heat: Temperature Dependent	x	x	x	x	o	x	x	x	x
Specific Heat: User Subroutine uspeci	x	x	x	x	o	x	x	x	x
Mechanical Properties									
Mechanical Type: Isotropy	x	x	x	x	x	x	x	x	o

Parameters	Modules *1								
	ST	TR/DP	FQ	LS	HT	EL-ST	EL-FQ	EL-CUR	DS
Mechanical Type: Orthotropy	x	x	x	x	x	x	x	x	o
Mechanical Type: Anisotropy (Matrix)	x	x	x	x	x	x	x	x	o
Density: Constant	x	o	x	x	o	x	x	x	o
Density: Temperature dependency	x	x	x	x	o	x	x	x	x
Density: User Subroutine udnsty	x	x	x	x	o	x	x	x	x
Young's Modules	o	o	o	x	x	x	x	x	o
Poisson's Ratio	o	o	o	x	x	x	x	x	o
Shear Modulus	o	o	o	x	x	x	x	x	o
Thermal Expansion Coefficient	x	x	x	x	x	x	x	x	o
Reference Temperature	x	x	x	x	x	x	x	x	o
Magnetostriction Type: Isotropic	o	o	o	x	x	x	x	x	x
Magnetostriction Type: Anisotropy	o	o	o	x	x	x	x	x	x
Piezoelectric Stress Constant	x	x	x	x	x	x	x	x	o
Loss									
Loss Type: Iron Loss Equation	x	x	x	o	x	x	x	x	x
Loss Type: Hysteresis/Joule Loss Table	x	x	x	o	x	x	x	x	x
Loss Type: Iron Loss Table	x	x	x	o	x	x	x	x	x

O: Available X: Not available

*1 Module abbreviations are as follows:

- TR: Magnetic field analysis (transient response analysis) *3D
- DP: Magnetic field analysis (transient response analysis) *2D, axisymmetric
- ST: Magnetic field analysis (static analysis) *3D, 2D, axisymmetric
- FQ: Magnetic field analysis (frequency response analysis) *3D, 2D, axisymmetric
- LS: Iron loss calculation *3D, 2D
- HT: Thermal analysis *3D
- EL-ST: Electric field analysis (static analysis) *3D
- EL-FQ: Electric field analysis (frequency response analysis) *3D
- EL-CUR: Current distribution analysis *3D
- CH: charge distribution analysis *3D
- DS: Structural analysis *3D, 2D

2-1-2. Material Database

The available materials that are registered in the material database for JMAG-Designer are indicated in the following table.

Parameters	Modules *1								
	ST	TR/DP	FQ	LS	HT	EL-ST	EL-FQ	EL-CUR	DS
Material Database									
Ambient	o	o	o	x	o	o	o	o	o
Permanent Magnet	o	o	x	x	x	x	x	x	o
Soft Magnetic Material	o	o	o	x	x	x	x	x	o
Conductor	o	o	o	x	o	o	o	o	o
Insulation	o	o	o	x	x	x	x	x	o

O: Available X: Not available

*1 Module abbreviations are as follows:

- TR: Magnetic field analysis (transient response analysis) *3D
- DP: Magnetic field analysis (transient response analysis) *2D, axisymmetric
- ST: Magnetic field analysis (static analysis) *3D, 2D, axisymmetric
- FQ: Magnetic field analysis (frequency response analysis) *3D, 2D, axisymmetric
- LS: Iron loss calculation *3D, 2D
- HT: Thermal analysis *3D
- EL-ST: Electric field analysis (static analysis) *3D
- EL-FQ: Electric field analysis (frequency response analysis) *3D
- EL-CUR: Current distribution analysis *3D
- CH: charge distribution analysis *3D
- DS: Structural analysis *3D, 2D

2-2. Magnetizing Properties of a Permanent Magnet

2-2-1. Important Notes Regarding Analysis for Using Permanent Magnet

- Permanent magnets cannot be used for magnetic field analysis (frequency response analysis).
- The initial magnetization for a permanent magnet can be confirmed during result display. Select the [Initial Magnetization] check box in the output settings for a magnetic field analysis when confirming.

▶▶ Reference

For more information about the output setting in a magnetic field analysis, see "3-52. Post Output" on page 316.

2-2-2. Types of Defining Method for Magnetic Properties

The magnets that can be used in JMAG are categorized into the following types.

- Linear
- Nonlinear (reversible)
- Nonlinear (reversible, temperature-dependent)
- Nonlinear (irreversible, thermal demagnetization, demagnetizing field)
- Nonlinear (irreversible, demagnetizing field)
- User subroutine (magusr)

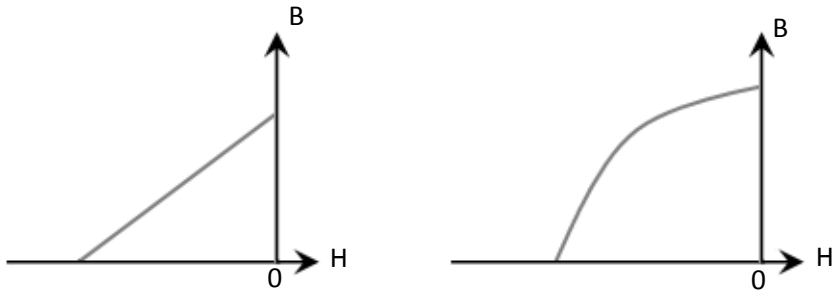
▶▶ Reference

For information about the reversible/irreversible of a magnet, see P.15 "2-2-3. Reversible and Irreversible Settings for a Permanent Magnet".

■ Linear

This type of magnet has linear magnetizing properties.

- User can define the magnetizing properties by specifying a constant relative permeability.



Magnetizing properties

■ Nonlinear (reversible)

This type of magnet has nonlinear magnetizing properties. The thermal demagnetization and alternating field demagnetization are not taken into account.

- User can define the magnetizing properties by creating a magnetic field - magnetic flux density curve.
- The database provided by the magnet manufacturers can be used (44 page).
  - The accurate results can be obtained because the magnetizing properties are based on the measurements taken at a specific temperature. However, the temperature other than the temperature that is used for the measurements cannot be selected.
  - If the temperature is not indicated, the measurements were taken at a normal temperature. For details, see the notes of each manufacturer.
- In JMAG-Studio, the B-H curve can be corrected (19 page).

▶▶ Reference

For the instructions on how to create the magnetic field - magnetic flux density curve, see P.78 "3-4-1. Specifying the Magnetization Direction of a Permanent Magnet".

■ Nonlinear (reversible, temperature-dependent)

This type of magnet has nonlinear magnetizing properties. The temperature can be specified for the magnet. The thermal demagnetization and alternating field demagnetization are not taken into account.

- User can define the magnetizing properties by creating a magnetic field- magnetic flux density curve at a specific temperature.
- The database provided by the magnet manufacturers can be used. The following magnets are available (44 page).
  - Hitachi Metals ferrite magnet
  - Hitachi Metals NdFeB magnet
- In JMAG-Designer, if multiple magnetic field - magnetic flux density curves that have different temperatures are specified for one material, the temperature can be specified for the magnet in the [Permanent Magnet] settings.
- In JMAG-Studio, the magnetization curve is approximated by the three lines. Based on the curve, a magnetization curve at the specified temperature is create in the program and used for an analysis. The temperature of the magnet can be specified by selecting constant or point sequence in the [Thermal Properties] tab of the [Material Editor] dialog box.

MEMO In a temperature-dependent magnetization curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

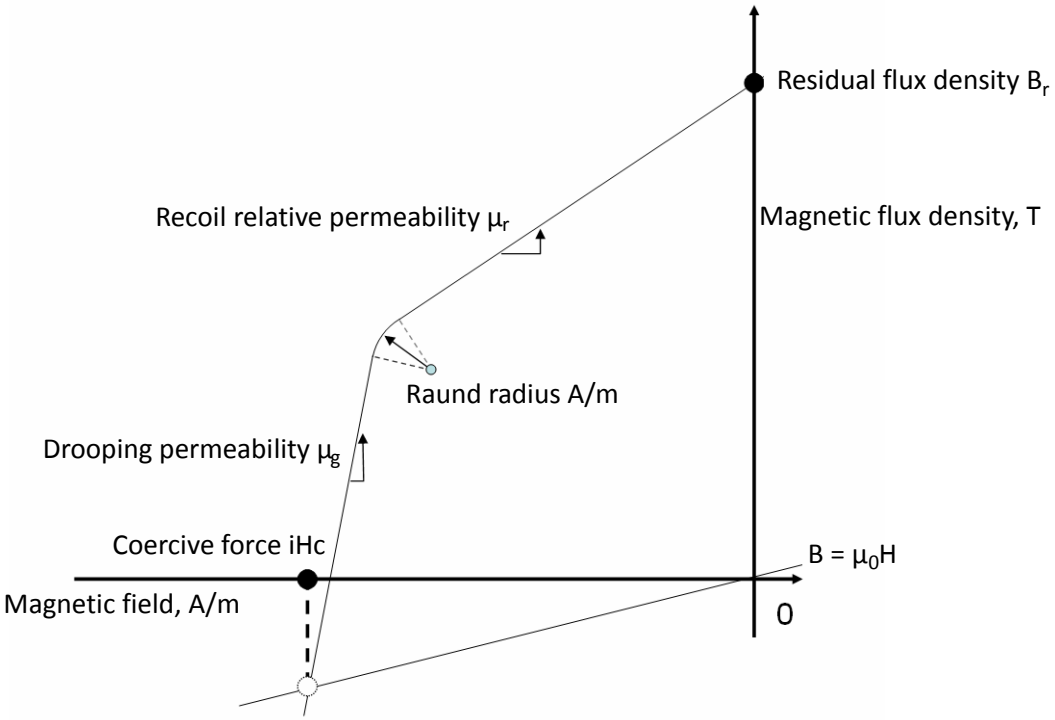
▶▶ Reference

For instructions on creating a magnetic field - magnetic flux density curve, see P.78 "3-4-1. Specifying the Magnetization Direction of a Permanent Magnet".

■ Nonlinear (irreversible, thermal demagnetization, demagnetizing field)

This type of magnet has nonlinear magnetizing properties. The thermal demagnetization and alternating field demagnetization where the performance of a magnet decays can be take into account.

- User can define the magnetizing properties by creating the temperature-dependent B-H curve. Specify temperatures in the [Temperature] column, and then specify [Coercive Force], [Flux Density], [Relative Permeability], and [Drooping Permeability] for each temperature.
  Specify a value in [Radius (A/m)] to display the changes in sudden curves in points more smoothly using curves. Specify the value zero in [Radius (A/m)] when using magnetic characteristics displayed from two forming lines. Specify the value zero in the radius when using magnetic characteristics displayed from two forming lines.



- In JMAG-Designer, a B-H curve can be created by specifying a point sequence.
- In JMAG-Studio, a HBP file contains a B-H curve is required.

▶▶ Reference

For instructions on how to use a HBP file, see “Setting User Magnet/User Steel”, “Material Properties: Magnetizing Properties” in Chapter 9, “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.

Important **The radius (A/m) for magnetization curve in temperature dependency can only be specified with magnets defined by the user. Material database provided by a manufacture of magnets cannot be specified.**

- The database provided by the magnet manufacturers can be used. The following magnets are available (44 page).
  - Hitachi Metals ferrite magnet
  - Hitachi Metals NdFeB magnet
- The magnetization state of an irreversible demagnetization magnet at the last step can be used as an initial value for a magnet in other analysis. For more information see, P.258 "3-31. Reusing Demagnetization (JMAG-Designer)".
- The magnetization curve is approximated by the three lines. Based on the curve, a magnetization curve at the specified temperature is create in the program and used for an analysis.
- The temperature of the magnet can be specified in the [Permanent Magnet] settings in JMAG-Designer and in the [Thermal Properties] tab of the [Material Editor] dialog box in JMAG-Studio.

MEMO In a temperature-dependent magnetization curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

■ Nonlinear (irreversible, demagnetizing field)

Magnet to display nonlinear characteristics for magnetic field. Performance deteriorated in a magnet by using a reverse magnetic field causing demagnetization can be used.

- User can define magnetic properties (Nonlinear/irreversible). Create a magnetic field - magnetic flux density curve. Specify also a recoil relative permeability.
- The database provided by the magnet manufacturers can be used. The following magnet is available (44 page).
  - Hitachi Metals AlNiCo magnet
- The magnetization state of an irreversible demagnetization magnet at the last step can be used as an initial value for a magnet in other analysis. For more information see, P.258 "3-31. Reusing Demagnetization (JMAG-Designer)".

■ User subroutine (magusr)

The magnetizing properties can be defined using the user subroutine *magusr* (345 page).

MEMO If the user subroutine is used, the correction feature for the magnetizing properties (93 page) cannot be used.

2-2-3. Reversible and Irreversible Settings for a Permanent Magnet

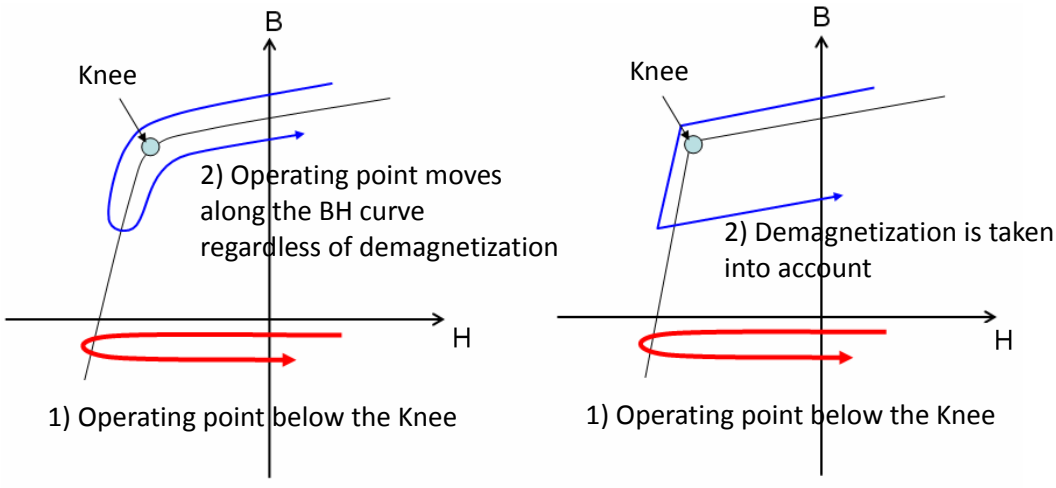
The reversible and irreversible settings for a permanent magnet are described below.

■ Reversible

When a nonlinear B-H curve is defined, the operating point moves along the B-H curve. Even if it goes beyond the knee of the curve during the analysis, demagnetization does not occur.

■ Irreversible

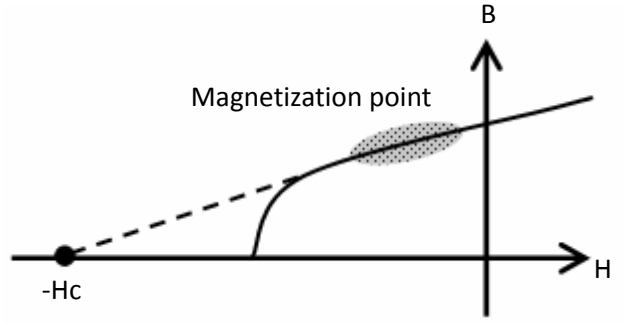
When a nonlinear B-H curve is defined, the operating point moves along the B-H curve. If it goes beyond the knee of the curve during the analysis, demagnetization occurs.



Reversible Irreversible

2-2-4. Specifying the Linear Magnetizing Properties of a Permanent Magnet

- Extrapolate the linear part of a demagnetization curve, and set the intersection of the extrapolated line and the magnetic field axis as the coercive force.
- Set the recoil relative permeability to be linear in the magnetizing properties.



Permanent magnet (linear)

2-2-5. Specifying the Nonlinear Magnetizing Properties of a Permanent Magnet

A magnetizing property curve can be created using a point sequence. When specifying the magnetizing property curve, note the following.

- It is desired that the magnetizing property curve represents the actual material quality as closely as possible. However, if the point sequence contains too many points, you may need to increase the number of nonlinear iterations.
- For a magnet, the demagnetization curve in the 2nd quadrant needs to be specified using a BH curve. The coercive force needs not to be specified.
- If there is a magnetic field that is higher than the maximum magnetic field defined by the magnetizing property curve, it is automatically interpolated based on the last point of the magnetizing property curve using the relative permeability in vacuum.
- Be sure to specify the magnetizing property curve point sequence in ascending order.

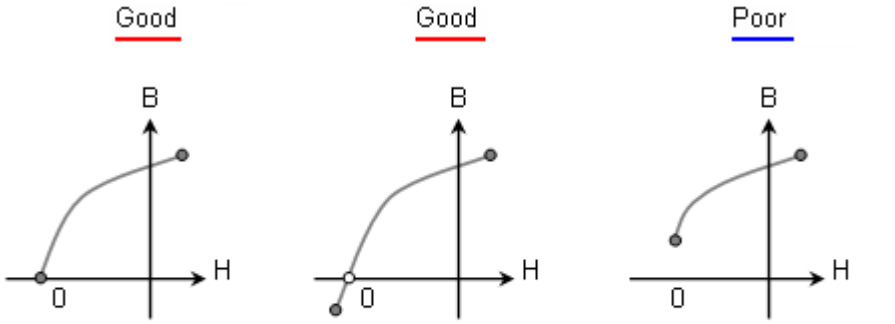
	Magnetic Field A/m	Flux Density T
1	-2000000	0
2	-1000000	0.5
3	-800000	1
4	-400000	2
5	0	2

	Magnetic Field A/m	Flux Density T
1	0	2
2	-400000	2
3	-800000	1
4	-1000000	0.5
5	-2000000	0

Good Example Poor Example

Examples: [Edit Point Sequence] dialog box (a B-H curve)

- Specify the point sequence so that the magnetizing property curve passes the point of *B=0*.



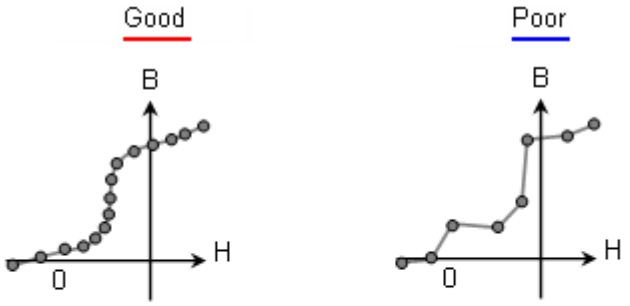
Start from the point of B=0

Passes the point of B=0

Does not pass the point of B=0

Examples

- Specify the point sequence so that the magnetizing property curve shows a smooth line.

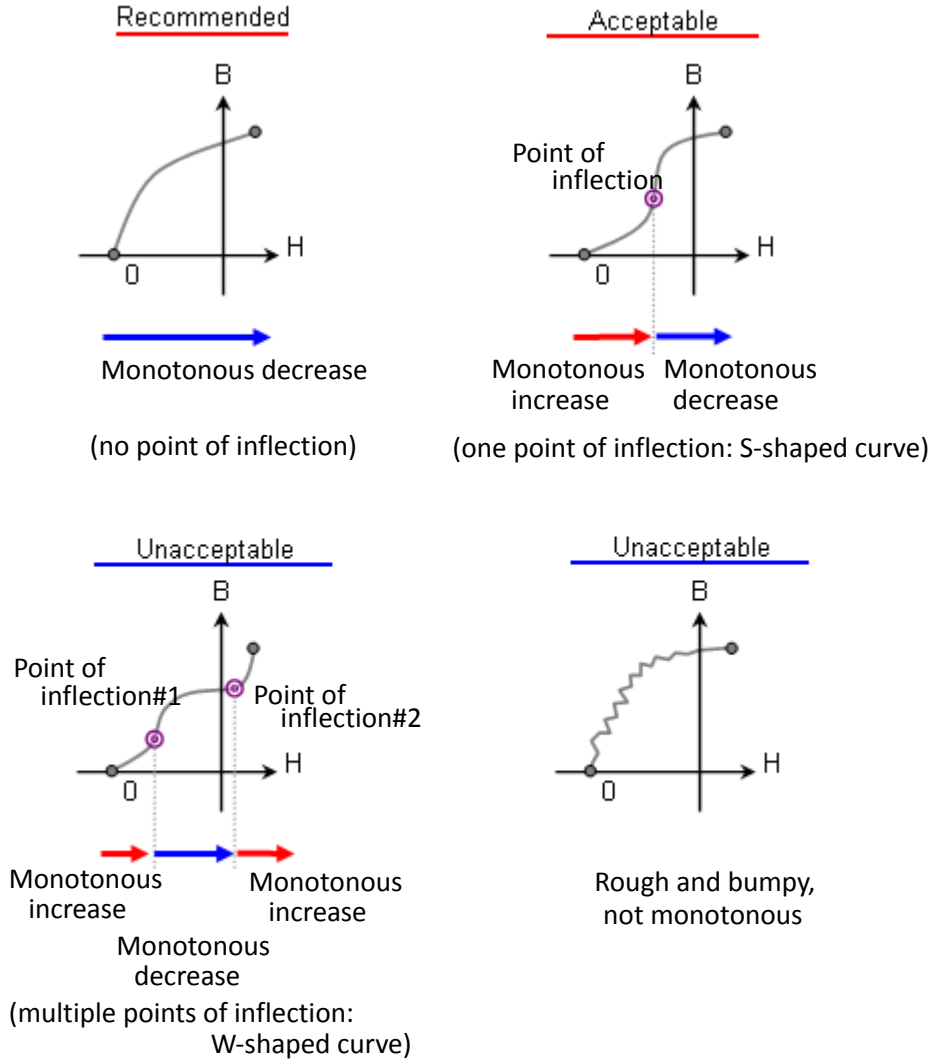


A smooth curve drawn with a large number of points

A rough curve drawn with a small number of points. The accuracy of the results is low.

Examples

- Specify the point sequence data so that the gradient of the magnetizing property curve monotonously decreases on the B-H plane. Be sure that there is not more than one point of inflection.
  When the gradient does not decrease monotonously (e.g., the cases of the "Acceptable" example below), the calculations may not converge sufficiently or may diverge. In such case, it is possible to improve the convergence by using Relaxation Factor. Note, however, that the calculation time may increase when this option is used.



Examples

2-2-6. Correcting the B-H Curve (JMAG-Studio)

When a B-H curve forms an S-shaped curve instead of a monotonously decreasing curve, the calculation may not converge sufficiently. In that case, correct the B-H curve.

■ Programming to modify the B-H point sequence

- The value of the *ith* magnetic field *H* is *H(i)* and the value of the *ith* flux density *B* is *B(i)*.
- The gradient of the B-H curve between the *ith* and *i+1th* is $\mu(i)$.
- The vacuum permeability is μ_0.

1. The value of $\mu'(i)$ at the *ith will be determined* using the *B-H* point sequence by applying *i* in sequential order starting from 1.

2. If (a) $\mu(i) < \mu(i+1)$, delete a point *(H(i+1), B(i+1))*,
or
if (b) $\mu < \mu_0$, delete a point *(H(i+1), B(i+1))*.

3. Return to i., and exit when both (a) and (b) are cleared.

MEMO A B-H curve can be corrected in the [Edit Point Sequence] dialog box, which can be opened from the [Point Sequence List] dialog box. To open the [Point Sequence List] dialog box, click [Settings] in the [Magnetizing Properties] tab in the [Material] dialog box, select [Nonlinear Magnet (B-H)] from the [Type] list in the [Magnetizing Properties] dialog box, and then click [Settings] under [Point Sequence].

2-3. Magnetizing Properties of a Soft Magnetic Material

2-3-1. Important Notes Regarding Analysis for Using Soft Magnetic Material

- The anisotropy can be set for the soft magnetic materials. The anisotropy can be set for the following types of magnetizing properties.
  - Linear (constant)
  - Nonlinear (B-H curve)

▶▶ Reference
For details, see P.97 "3-5-2. Specifying Lamination Direction for Magnetic Steel".

- The complex relative permeability can be specified for the magnetizing properties in frequency response analysis (FQ). The complex relative permeability represents the phase delay of the magnetic flux density in the alternating magnetic field. The real part is associated with the same phase component as the magnetic field of the magnetic flux density, and the imaginary part is associated with hysteresis loss which is calculated in frequency response analysis. The hysteresis loss is calculated in a frequency response analysis as an output item.
- The negative value needs to be specified for the imaginary part to set the complex relative permeability. If a positive value is specified for the imaginary part, the hysteresis loss is exported in the negative value.
- The permeability of parts can be fixed and the material properties can be defined from the magnetic flux density and magnetization curve by using the frozen permeability condition (251 page) in JMAG-Designer.

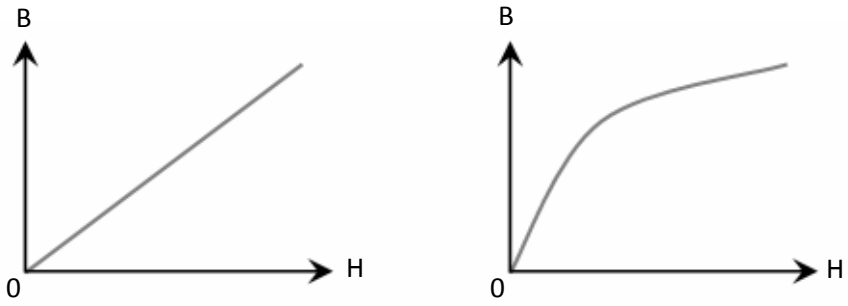
2-3-2. Types of Defining Method for Magnetizing Properties

The soft magnetic materials that are used in JMAG are grouped into the following categories.

- Linear (constant)
- Linear (temperature dependent (T-μ curve))
  * For coupling with magnetic field and thermal analysis
- Nonlinear (B-H curve)
- Nonlinear (B-M curve)
- Nonlinear (B-μ curve)
- Nonlinear (B-H curve (Stress-dependent))
- User subroutine (magusr)
- Nonlinear (B-H curve (Temperature dependent))
  * Can only be used for a coupled magnetic and thermal analysis and JMAG-Designer

■ Linear (constant)

This type of soft magnetic material has linear magnetizing properties. The relative permeability is constant.



Linear Nonlinear

Magnetizing properties

■ Linear (temperature dependent (T-μ curve))

This property is used for a coupled magnetic field and thermal analysis.

A soft magnetic material which permeability varies with temperature. Define the relative permeability versus temperature curve, and run a couple magnetic field and thermal analysis.

MEMO In a relative permeability-temperature curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

MEMO The temperature distribution condition (255 page) needs to be set when using soft magnetic material with temperature-dependent magnetic property for JMAG-Designer.

▶▶ Reference

For more information about coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

■ Nonlinear (B-H curve, B-M curve, B-μ curve)

A soft magnetic material has nonlinear magnetizing properties.

- User can define the compressive stress. Create the one of the following magnetizing properties.
  - B-H curve (magnetic field - magnetic flux density curve)
  - B-M curve (magnetic flux density - magnetization curve)
  - B-μ curve (magnetic flux density - relative permeability curve)

  MEMO The correction feature (98 page) cannot be used for the B-μ curve.
- The database provided by the steel sheet manufacturers can be used (44 page). If a material in the database is used, the direct current (DC) magnetizing properties defined in the database is used as the B-H curve of the material.
- In JMAG-Studio, the B-H curve can be corrected (19 page).

▶▶ Reference

For instructions on how to create a B-H curve, see P.25 "2-3-3. Nonlinear Magnetizing Properties for a Soft Magnetic Material".

■ Nonlinear (B-H curve (stress-dependent))

This type of soft magnetic material has nonlinear magnetizing properties. The compressive stress can be set.

- User can define the magnetizing properties. Multiple B-H curves at a specific stress need to be created.
  - In JMAG-Designer, the magnetizing properties can be created using a point sequence.
  - In JMAG-Studio, a CSV file contains the magnetizing properties is required.

▶▶ Reference

For instructions on how to use a CSV file, see “Setting User Magnet/User Steel”, “Material Properties: Magnetizing Properties” in Chapter 9, “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.

MEMO The stress distribution condition (253 page) needs to be set when using soft magnetic material with stress-dependent magnetic property.

▶▶ Reference

For more information, see P.29 "2-3-5. Specifying the Magnetic Properties of a Stress Dependency".

■ User subroutine (magusr)

The magnetizing properties can be defined using the user subroutine *magusr* (345 page).

MEMO If the user subroutine is used, the correction feature for the magnetizing properties (98 page) cannot be used.

■ Nonlinear (B-H curve (temperature-dependent)) (JMAG-Designer)

This property is used for a coupled magnetic field and thermal analysis.

This is a soft magnetic material which changes by temperature and specifies a non-linear characteristic for a magnetic field. Create a base temperature for B-H curve (magnetic field - magnetic flux density curve) and a table for temperature and magnetic susceptibility. A temperature is determined by coupling with a thermal analysis.

The base temperature is k=10 that is based on the table for the temperature and magnetic susceptibility. [k=1.0] needs to be specified in the table for the temperature and magnetic susceptibility.

The following process is used when calculating the temperature - magnetic susceptibility from a B-H curve of each temperature. The last point of each B-H curve needs to reach the saturation area of a region.

i. The magnetization M(T) is calculated from the last point of the B-H curve for temperature T.

ii. The magnetic susceptibility [k=M(T) / $M(T_0)$] for each temperature is calculated by using magnetization $M(T_0)$ from the base temperature T_0.

MEMO The temperature distribution condition (255 page) needs to be set when using magnetic material with temperature-dependent magnetic property for JMAG-Designer.

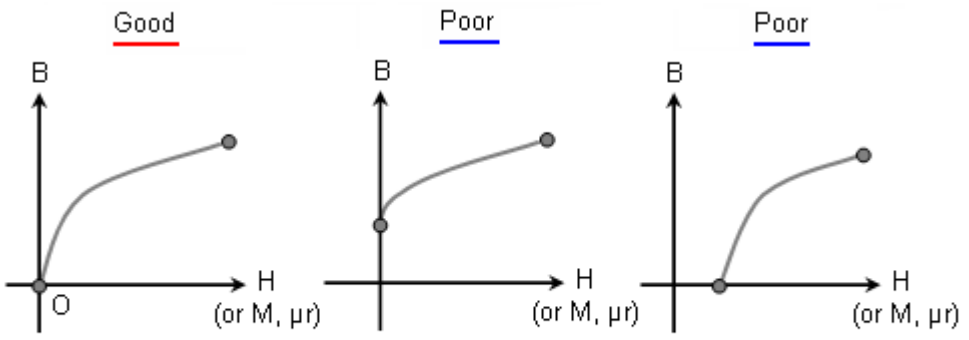
▶▶ Reference

- For more information, see P.30 "2-3-6. Specifying the Magnetic Properties of a Temperature Dependency".
- For more information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

2-3-3. Nonlinear Magnetizing Properties for a Soft Magnetic Material

Note the following when creating nonlinear magnetizing properties (B-H curve, B-M curve, B-μ curve).

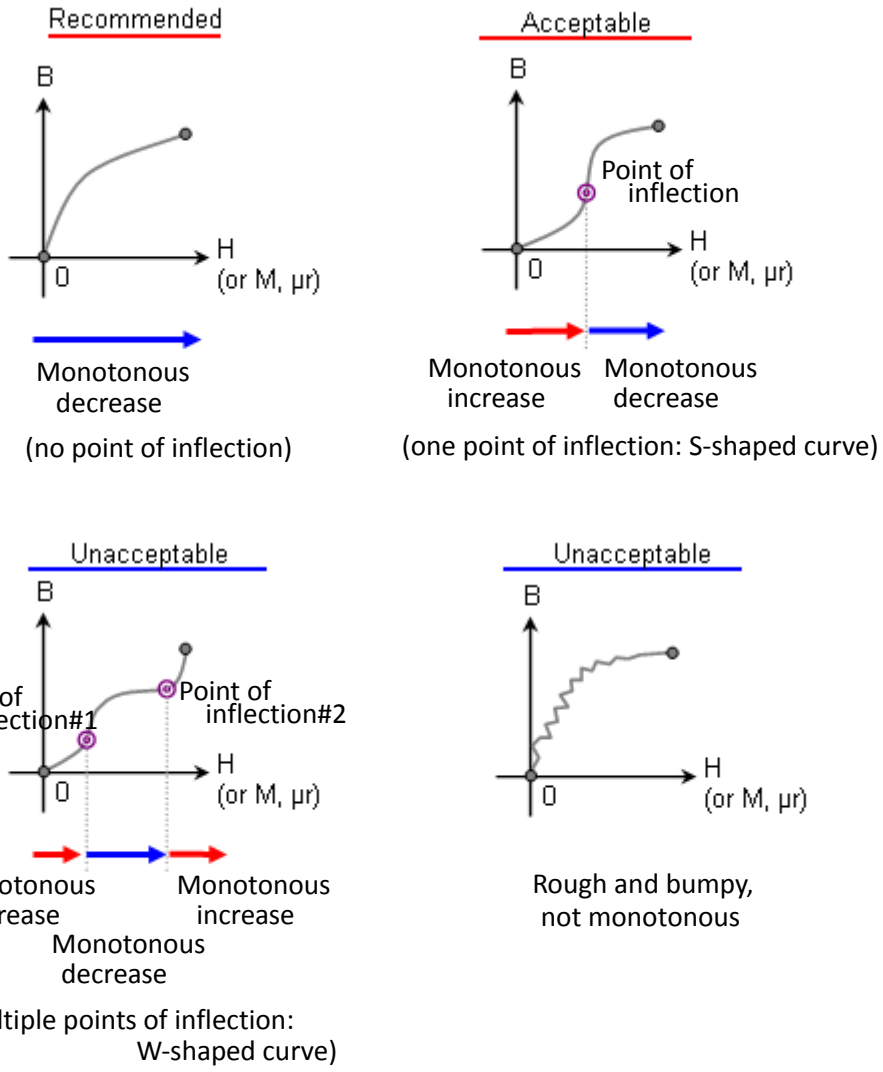
- It is desirable to have a magnetizing property curve representing the material as accurately as possible. However, if the point sequence contains too many points, the number of nonlinear iterations may need to be increased.
- If there is a magnetic field that is higher than the maximum specified for the magnetizing property curve, it is automatically interpolated from the last point of the magnetizing property curve with the relative permeability in vacuum.
- Specify the point sequence data of the magnetizing property curve in ascending order.
- Specify the point sequence data so that the magnetizing property curve passes through the point *B=0*.
- For a nonlinear material with Bμ curve, the relative permeability μr is the gradient determined by the origin and a point on the B-H curve.
- Specify the first point to (0,0) for the B-H curve and BM curve, as well as the user-defined steel of JMAG-Studio or JMAG-Designer.



Examples

- Specify the point sequence data so that the magnetizing property curve becomes a smooth line.

- Specify the point sequence data so that the gradient of the magnetizing property curve decreases monotonously on the B-H plane. Be sure that there is not more than one point of inflection.
  If the gradient does not decrease monotonously (e.g. the cases "Acceptable" below), the calculations may not converge sufficiently or may diverge. In such case, it is possible to improve the convergence by using Relaxation Factor. Note, however, that the calculation time may increase when this option is used.



Examples

- In the frequency response analysis, either of the following two methods can be used for the nonlinear magnetizing properties.
  - Do not correct the nonlinear magnetization curve
    The amount of magnetic field and the amount of magnetic flux density can be related using the nonlinear magnetization curve. The accurate results of electromagnetic force, torque, and Joule loss cannot be obtained because magnetic saturation is not taken into account.
  - Correct the nonlinear magnetization curve
    If the magnetic field changes sinusoidally, the magnetic flux density does not change sinusoidally due to magnetic saturation. To include the effect of magnetic saturation, correct the nonlinear magnetization curve. This method is used to obtain the accurate results of electromagnetic force, torque, and Joule loss.

Reference

To use the correction feature, select [Use Correct BH Curve for Frequency Response Analysis]. For details, see P.315 "3-51-2-6. Correcting the B-H Curve for Frequency Response Analysis".

2-3-4. Magnetizing Properties for an Anisotropic Magnetic Steel Sheet

For an anisotropic magnetic steel sheet, it is possible to set different magnetizing properties for the easy-axis direction, hard-axis direction, and lamination direction.

- Be sure to set the magnetizing properties in the easy-axis direction.
- Magnetic flux does not flow in the lamination direction well. Specify the lamination factor to adjust the magnetizing properties in the lamination direction. If not adjusting the properties, set the lamination factor to 100%.
  If specifying the lamination factor but not specifying the magnetizing properties in the lamination direction, the properties are determined based on the magnetizing properties in the easy-axis direction.
- The components of flux density are calculated based on the magnetizing properties in each direction. The relationship between the flux density and the magnetic field can be expressed as follows.

$$H = (H_x, H_y, H_z), H_x = H_x(B_x), H_y = H_y(B_y), H_z = H_z(B_z)$$

When the magnetic field is generated in a direction other than the specified three directions, the accuracy of the calculation decreases. (The total of the flux density becomes large.).

- The following are the eight ways to determine the three directions. (1), (2), (3), (4), and (7) are available in 2D analysis, 3D analysis, and axisymmetric analysis. (5), (6), and (8) are only available in 3D analysis.

Type		Description
(1)	Orthogonal coordinate system	Determines the local coordinate system (X-axis: easy-axis direction, Y-axis: hard-axis direction, Z-axis: lamination direction)
(2)	User subroutine	Determines the local coordinate system using a user subroutine *usrmtx* (371 page).
(3)	Cylindrical coordinate system	Determines the cylindrical coordinate system (r-direction: lamination direction, θ-direction: easy-axis direction, Z-direction: hard-axis direction)
(4)	Cylindrical coordinate system	Determines the cylindrical coordinate system (r-direction: easy-axis direction, θ-direction: lamination direction, Z-direction: hard-axis direction)
(5)	Cylindrical coordinate system	Determines the cylindrical coordinate system (r-direction: lamination direction, θ-direction: hard-axis direction, Z-direction: easy-axis direction)
(6)	Cylindrical coordinate system	Determines the cylindrical coordinate system (r-direction: hard-axis direction, θ-direction: lamination direction, Z-direction: easy-axis direction
(7)	Cylindrical coordinate system	Determines the cylindrical coordinate system (r-direction: hard-axis direction, θ-direction: easy-axis direction, Z-direction: lamination direction)
(8)	Cylindrical coordinate system	Determines the cylindrical coordinate system (r-direction: easy-axis direction, θ-direction: hard-axis direction, Z-direction: lamination direction)

2-3-5. Specifying the Magnetic Properties of a Stress Dependency

- Magnetizing properties with compressive stress set can be used for an analysis. The following settings are necessary.
  - Select the stress-dependent magnetizing properties.
  - Set the stress distribution condition (253 page).

■ Materials with magnetic properties (stress dependent)

- The following types of stress-dependent magnetizing properties are available.
  - The user creates the magnetic materials
    - In JMAG-Designer, the stress-dependent magnetizing properties can be defined in the [Material Editor] dialog box.
    - In JMAG-Studio, the stress-dependent magnetizing properties can be defined by setting [User_Steel] from the [Type] list in the [Magnetizing Properties] dialog box, and then importing the CSV file that contains the stress dependent magnetizing properties curve.

      ▶▶ Reference

      For instructions on how to create a CSV file, see “Setting User Magnet/User Steel”, “Material Properties: Magnetizing Properties” in Chapter 9, “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.

■ Referred stress

- The stress distribution can be set in the following two ways.
  - Specify the stress distribution for each material
  - Use the stress distribution obtained from the structural analysis (static analysis)
  - Run a two-way coupled magnetic field and structural analysis

  If specifying the stress distribution for each material, use compressive stress (negative value).

  If using the stress distribution of structural analysis, reference the compressive component of the principal stress. To include all components of the stress tensor, use a user subroutine *usrstr* (386 page).
- If using a user-defined steel sheet, the defined stress values are linearly interpolated. If a value outside the defined range is referenced, the value at the end of the range will be used. If not using linear interpolation, a user subroutine *usrstr* is used.
- The principal stress to reference can be confirmed in the output results.

▶▶ Reference

For more information about coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

2-3-6. Specifying the Magnetic Properties of a Temperature Dependency

The magnetic properties using the temperature exported from a thermal analysis can be used for an analysis. The following settings need to be specified when using this function.

- Setting materials with magnetic properties (temperature dependency) to a part.
- Setting the temperature distribution condition (255 page) *JMAG-Designer only

■ Materials with magnetic properties (temperature dependent)

- The types of materials with magnetic properties (temperature dependent) are as follows.
  - The user creates the magnetic materials
    - Materials with magnetic properties of temperature dependency can be created in the [Material Editor] dialog in JMAG-Designer.

      ▶▶ Reference

      For more information, see "■Linear (temperature dependent (T-m curve))" on page 22, "■Nonlinear (B-H curve (temperature-dependent)) (JMAG-Designer)" on page 24.
    - Select [Linear Material (T-μ)] in the [Type] list of the [Magnetizing Properties] dialog box to define a curve for temperature - relative permeability in JMAG-Studio.

■ Referred temperature

- The following two types of methods specify temperature distribution.
  - Refers to a result file exported by a thermal analysis and imports a temperature distribution (one way coupled analysis)
  - Run a two-way coupled magnetic field and thermal analysis

    ▶▶ Reference

    For more information about coupled analysis, see "Chapter 10. Coupled Analysis" on page 769.

2-4. Electric Properties

2-4-1. Specifying Electrical Conductivity/Electric Resistivity (Magnetic Field Analysis)

In the following cases, the constant electrical conductivity or constant electric resistivity needs to be specified.

- When an FEM conductor is used (141 page).
- When eddy current is to be calculated (99 page).
- When the Joule loss of a part with the current condition (113 page) set is to be obtained.

The following should be noted when setting the electric conductivity and electric resistivity.

- The electric conductivity needs to be specified for the conductor when an FEM conductor is used.

  In a 3D analysis, the inflow face and outflow face need to be specified for the FEM conductor condition. How the current flows is determined by the electric conductivity that is specified for the part.
- The Lorentz force density and Joule loss density can be calculated for the parts where the electric conductivity is specified. Whether or not to export the results can be selected.
- The resistivity is already defined for some materials in the material database provided by the manufactures (44 page).
- The electric conductivity can also be defined using a user subroutine *conusr* (349 page).

Important **If an FEM coil is used, the electric conductivity should not be specified.**

2-4-2. Specifying Electric Conductivity/Electric Resistivity (Electric Field Analysis, Charge Distribution Analysis)

- Specify this setting for an electric field analysis (frequency response analysis) and a current distribution analysis.
- A characteristic can be defined in constant electric conductivity/electric resistivity when a material has linear characteristics for electric fields with electric conductivity/ electric resistance.
- A characteristic can be defined in point sequence data (electric conductivity/electric resistivity) when a material has nonlinear characteristics for electric fields with electric conductivity/electric resistivity. The value "0.0" cannot be specified in electric conductivity/electric resistivity.
- A characteristic can be defined in point sequence data (electric conductivity/electric resistivity) when a material has nonlinear characteristics for temperature with electric conductivity/electric resistivity. The value "0.0" cannot be specified in electric conductivity/electric resistivity.

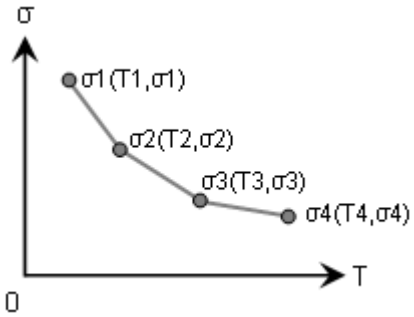
  ▶▶ Reference

  For more information, see "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)" on page 33.
- A characteristic can be defined in point sequence data (electric conductivity/electric resistivity - frequency dependency) when a material has nonlinear characteristics for frequencies with electric conductivity/ electric resistivity. The value "0.0" cannot be specified in electric conductivity/electric resistivity.
- The matrix can be defined by using the user subroutine *cndusr* (543 page) for a material with electric conductivity. Specify the ID (negative integer) for the argument iflag when using *cndusr*.

2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)

Defines the characteristics by using one of the following point sequence and linking to a thermal analysis when the electric conductivity or electric resistivity of a material that is used for an analysis is going to be temperature dependent.

- T-σ curve (temperature - electrical conductivity curve)
- T-p curve (temperature - electrical resistivity curve)



Temperature-dependent electrical conductivity (T-σ curve)

MEMO In a temperature - electric conductivity curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

▶▶ Reference

For information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

2-4-4. Specifying Anisotropy for the Electric Properties (JMAG-Studio)

In JMAG-Studio, the anisotropy can be set for the electrical conductivity of the materials.

- This anisotropy can be set for non-magnetic materials, isotropic magnetic materials, and magnet materials.
  - After specifying the 1st, 2nd and 3rd axis directions for the electric properties, define the electrical conductivity of each axis.
  - The electrical conductivity can be defined in the following two ways.
    - Constant
      Specifies the electrical conductivity.
    - Temperature function
      Defines the temperature - electrical conductivity (T-σ) curve using a point sequence.

    MEMO The method for defining the electric conductivity needs not to be same for all the axis directions.

MEMO For isotropic magnetic steel sheets and anisotropic magnetic steel sheets, the setting of the direction of anisotropy (lamination direction) of the magnetizing properties is used to define the lamination direction of the electric properties.

2-4-5. Specifying Relative Permittivity (Electric Field Analysis, Charge Distribution Analysis)

- Specify this setting for an electric field analysis (static analysis, frequency response analysis) and a charge distribution analysis.
- A characteristic can be defined by a constant relative permittivity when a material permittivity has linear characteristics for electric fields. Specify the value for a real part and a imaginary part when running a frequency response analysis.
- A characteristic can be defined by a point sequence (electric field - relative permittivity) when a material permittivity has nonlinear characteristics for electric fields. Specify the value for a real part and an imaginary part when running a frequency response analysis. The value "0.0" cannot be specified as a relative permittivity.
- A characteristic can be defined by a point sequence (frequency - relative permittivity) when a material permittivity has nonlinear characteristics for frequency. Specify the value for a real part and a imaginary part when running a frequency response analysis. The value "0.0" cannot be specified for a relative permittivity.

2-4-6. Specifying Permittivity Settings (Structural Analysis)

- Specify each component of the relative permittivity matrix when running a structural analysis for a piezoelectric analysis in each study (static analysis, eigenmode analysis, frequency response analysis).
- The relative permittivity is defined by the following formula.

$$\{D\} = [e]\{S\} - [e^S]\{E\}$$

$$[e^S] = \begin{bmatrix} E_{11} & & \\ & E_{22} & \\ & & E_{33} \end{bmatrix}$$

[eS]: Permittivity matrix [F/m]

[e]: Piezoelectric stress constant matrix e [C/m^2]

{D}: Electric displacement vector [C/m^2]

{S}: Strain vector

{E}: Vector for electric field intensity [N/C]

MEMO Piezoelectric analysis is supported for a 3D analysis.

MEMO Specification of the piezoelectric stress constant, relative permittivity, local coordinate of each part, and the electric potential boundary condition needs to be set when running a piezoelectric analysis.

▶▶ Reference

- For information about piezoelectric stress constant, see "2-4-6. Specifying Permittivity Settings (Structural Analysis)" on page 35.
- For information about the piezoelectric analysis and the electric potential boundary condition, see "6-6. Electric Potential Boundary (Piezoelectric analysis)" on page 584.

2-5. Thermal Properties

2-5-1. Specifying Thermal Conductivity

- A constant thermal conductivity needs to be specified if thermal conductivity of a material does not vary with temperature.
- Nonlinear material properties need to be set using a point sequence if thermal conductivity of a material varies with temperature.

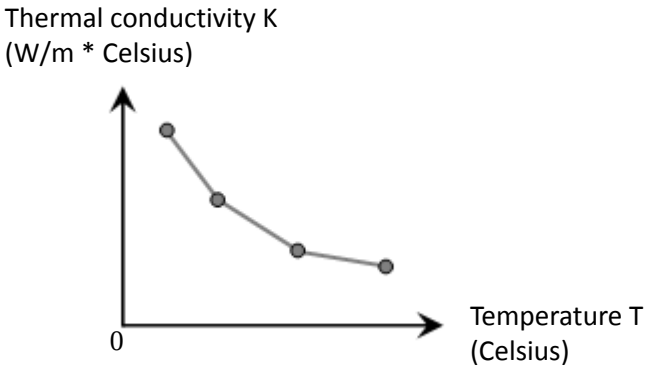


Fig.1. Temperature - thermal conductivity curve

- A temperature - thermal conductivity curve is used to indicate the nonlinear material properties as shown in Fig. 1.
  In a temperature - thermal conductivity curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.
- An anisotropic temperature - thermal conductivity curve can be obtained using a user subroutine *ucond* (478 page).
- If a negative value (e.g. -1.0000) is specified for the thermal conductivity of a material, the material is not analyzed.

2-5-2. Specifying Specific Heat

- A specific heat needs to be specified for a transient state analysis.
- A constant specific heat needs to be specified if specific heat of a material does not vary with temperature.
- Nonlinear material properties need to be set using a point sequence if specific heat of a material varies with temperature.

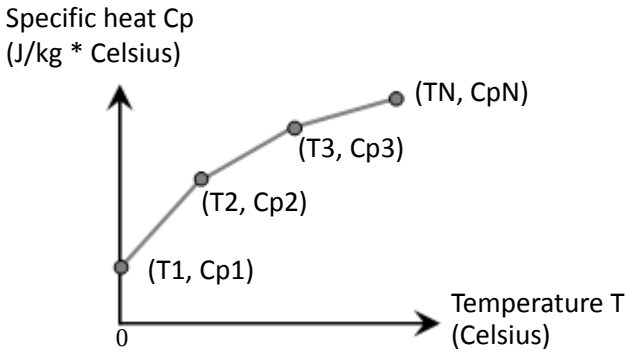


Fig.2 Temperature - specific heat capacity curve

- A temperature - specific heat curve is used to indicate the nonlinear material properties as shown in Fig. 2.
  In a temperature - specific heat curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.
- An anisotropic temperature - specific heat curve can be obtained using the user subroutine *uspeci* (480 page).

2-6. Mechanical Properties

2-6-1. Specifying the Density (Magnetic Field Analysis)

The mass density of material is used for calculating the mass or moment of inertia which are then used as parameters for the equation of motion in the motion condition settings.

If using the material database (44 page), the mass density specific to the material will be displayed.

▶▶ Reference

For information about equation of motion in the motion condition, see "3-17-2-5. Equation of Motion" on page 189.

2-6-2. Specifying the Density (Thermal Analysis)

- The density needs to be specified for a transient state analysis.
- A constant density needs to be specified if the density of a material varies with temperature.
- The temperature density curve can be used as a point sequence data to show nonlinear characteristics.
  In a temperature - density curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.
- The density can be defined by using the user subroutine *udnsty* (493 page).

2-6-3. Specifying the Magnetostriction (Magnetic Field Analysis)

- The magnetostriction properties for materials are set when calculating magnetostrictive force in a magnetic field analysis. The magnetostrictive force calculated from a magnetic field analysis is used in a structural analysis.
- The magnetic flux density-strain curve (B- ε curve) is specified when a material has magnetostriction properties that are isotropic.
- The magnetic flux density-strain curve (B- ε curve) for a rolling direction and a transverse direction are specified, when materials with magnetostriction properties are anisotropy.
  A specified lamination pattern is referenced, in order to determine a rolling direction and a transverse direction. This occurs when magnetostriction properties that are anisotropy are specified.

MEMO A laminated pattern specified in a magnetic field analysis is also applied during a structural analysis, when using magnetostrictive force calculated from a magnetic field analysis in a structural analysis.

▶▶ Reference

For more information about lamination patterns, see "3-5-2. Specifying Lamination Direction for Magnetic Steel" on page 97.

- The magnetic flux density-strain curve (B-ε curve) are linearly interpolated.
- Both ends on a B-H curve are used for extrapolating, when a value outside the specified area in a magnetic flux density-strain curve (B- ε curve) during an analysis is referenced.

▶▶ Reference

For more information about calculating magnetostrictive force, see "3-23. Magnetostriction Calculation (JMAG-Designer)" on page 232.

2-6-4. Specifying Mechanical Properties (Isotropic/Anisotropy) (Magnetic Field Analysis)

- Mechanical properties need to be set when calculating magnetostrictive force in a magnetic field analysis. Specify the following settings.
  - Mechanical properties (isotropic/anisotropy)
  - Young's modulus (PA), (KPa), (MPa)
  - Poisson's ratio
  - Shear modulus

▶▶ Reference

For more information about calculating magnetostrictive force, see "3-23. Magnetostriction Calculation (JMAG-Designer)" on page 232.

2-6-5. Specifying the Mechanical Properties (Isotropic) (Structural Analysis)

- Specify the following settings.
  - Density (kg/m^3), (g/m^3), (g/cm^3), (lb/ft^3), (lb/in^3)
  - Young's modules (Pa), (KPa), (MPa)
  - Shear modulus (Pa), (KPa), (MPa)
  - Poisson's ratio
  - Thermal expansion coefficient (1/Celsius), (1/Fahrenheit)
  - Reference temperature (Celsius), (Fahrenheit)
- The following should be noted when using this function.
  - A thermal load is calculated based on the reference temperature set for the material properties and the nodal temperature set for the temperature load condition.
  - The reference temperature is a temperature used to define the thermal expansion coefficient. If the reference temperature and the nodal temperature are different, the thermal load is caused depending on the thermal expansion coefficient, and a part is deformed.
    For example, if the nodal temperature is zero degrees Celsius and the reference temperature is 20 degrees Celsius, the thermal load of -20 degrees Celsius is applied to the node.
  - The load is not applied to areas where the temperature load condition is not set.

2-6-6. Specifying the Mechanical Properties (Anisotropy) (Structural Analysis)

- Specify the following items for anisotropy (Matrix).
  - Density (kg/m^3), (g/m^3), (g/cm^3), (lb/ft^3), (lb/in^3)
  - Young's Modulus (Pa), (KPa), (MPa)
  - Shear Modulus (Pa), (KPa), (MPa)
  - Poisson's ratio
  - Thermal expansion (1/Celsius), (1/°F)
  - Reference temperature (Celsius), (°F)
  - Anisotropy (Matrix)
- Specify the following items for orthotropy (Young's modulus or Poisson's ratio).
  - Density (kg/m^3), (g/m^3), (g/cm^3), (lb/ft^3), (lb/in^3)
  - Matrix
  - Thermal expansion (1/Celsius), (1/°F)
  - Reference temperature (Celsius), (°F)

- The following should be noted when using this function.
  - The thermal load is calculated with the reference temperature of the material properties and the nodal temperature set in the temperature load condition.
  - The reference temperature explained in this section is the temperature that is the basis that is defined in the thermal expansion. A thermal load occurs and a part is deformed in response to the thermal expansion when there is a difference in the reference temperature and nodal temperature. For example, if the nodal temperature is 0 degrees and the reference temperature is 20 degrees, the load causes stress, deformations and displacements in -20 degrees.
  - A load is not created in an area where a temperature load condition is not set.
  - The rectangular coordinate system and the cylindrical coordinate system can be used for the material coordinate system.
  - Specify each components for the matrix displayed in the following formula when [Anisotropy (Matrix)] is set.
    - Second dimension

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 \\ G12 & G22 & G23 \\ G13 & G23 & G33 \end{bmatrix} \left[\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \Upsilon_{xy} \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} A_x \\ A_y \\ A_{xy} \end{Bmatrix} \right]$$

      [σ]: Stress, [Gij]: Rigid matrix, [ε]: Strain, T: Nodal temperature, T_{ref}: Reference temperature, [A]: Thermal expansion vector, x, y, xy: Each component of the coordinate system applied to a material
    - Third Dimension

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 & G14 & G15 & G16 \\ & G22 & G23 & G24 & G25 & G26 \\ & & G33 & G34 & G35 & G36 \\ & & & G44 & G45 & G46 \\ & symmetric & & & G55 & G56 \\ & & & & & G66 \end{bmatrix} \left[\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \Upsilon_{xy} \\ \Upsilon_{yz} \\ \Upsilon_{zx} \end{Bmatrix} - \begin{Bmatrix} A_x \\ A_y \\ A_z \\ A_{xy} \\ A_{yz} \\ A_{zx} \end{Bmatrix} (T - T_{ref}) \right]$$

      [σ]: Stress, [Gij]: Rigid matrix, [ε]: Strain, T: Nodal temperature, T_{ref}: Reference temperature, [A]: Thermal expansion vector, x, y, z, xy, yz, zx: Each component of the coordinate system applied to a material

2-6-7. Specifying the Piezoelectric Stress Constant (Structural Analysis)

- Specify each component of the piezoelectric stress constant matrix when running a piezoelectric analysis for each study (static analysis, eigenmode analysis, frequency response analysis) in a structural analysis.
- The piezoelectric stress constant is specified when the mechanical properties (anisotropy) is set. This function is not supported by the mechanical properties (isotropic)
- The piezoelectric stress constant is defined by the following formula.

$$\{T\} = [C^E]\{S\} - [e]^T\{E\}$$

[e]: Piezoelectric stress constant matrix e [C/m^2]

[CE]: Elastic stiffness matrix at constant electric field [Pa]

{T}: Stress vector [Pa]

{S}: Strain vector

{E}: Vector for electric field intensity [N/C]

- The piezoelectric constant may be applied as 'd geometry' in a material data.

$$[d] = \begin{bmatrix} d_{11} & d_{12} & d_{13} & d_{14} & d_{15} & d_{16} \\ d_{21} & d_{22} & d_{23} & d_{24} & d_{25} & d_{26} \\ d_{31} & d_{32} & d_{33} & d_{34} & d_{35} & d_{36} \end{bmatrix}$$

JMAG uses e matrix where it needs to be converted in the following formula.

$$[e] = [d][C^E]$$

$$[e]^T = \begin{bmatrix} e_{11} & e_{12} & e_{13} & e_{14} & e_{15} & e_{16} \\ e_{21} & e_{22} & e_{23} & e_{24} & e_{25} & e_{26} \\ e_{31} & e_{32} & e_{33} & e_{34} & e_{35} & e_{36} \end{bmatrix}^T = \begin{bmatrix} e_{11} & e_{21} & e_{31} \\ e_{12} & e_{22} & e_{32} \\ e_{13} & e_{23} & e_{33} \\ e_{14} & e_{24} & e_{34} \\ e_{15} & e_{25} & e_{35} \\ e_{16} & e_{26} & e_{36} \end{bmatrix}$$

MEMO Piezoelectric analysis is supported.

MEMO Specification of the piezoelectric stress constant, relative permittivity, local coordinate of each part, and the electric potential boundary condition needs to be set when running a piezoelectric analysis.

Reference

- For information about relative permittivity, see "2-4-6. Specifying Permittivity Settings (Structural Analysis)" on page 35.
- For information about the piezoelectric analysis and the electric potential boundary condition, see "6-6. Electric Potential Boundary (Piezoelectric analysis)" on page 584.

2-7. Material Database

2-7-1. Function

JMAG offers the material database containing extensive material data made available by the manufacturers. The following should be noted for using the material database.

- If a magnetic field that is higher than the maximum magnetic field defined in the material database, it is automatically interpolated based on the last point of the B-H curve using the relative permeability in vacuum.
- When the magnet database is used, the values of coercive force and the demagnetization curve in the database are used.
- Some of the magnets in the database are temperature-dependent, and a B-H curve at a specific temperature can be selected for an analysis.
- For some of the magnets in the database, the temperature is automatically interpolated during an analysis by approximating the demagnetization curve with three straight lines.

  The magnetizing properties may change during an analysis as the demagnetization due to the temperature variations and external magnetic field. When the demagnetization due to temperature variations is taken into account, the temperature variations need to be set for the material properties, or run a coupled magnetic field and thermal analysis. The magnets of Hitachi Metals and the user-defined magnets are supported.
- When the magnetic steel sheet database is used, the direct current magnetization properties in the database are used for the B-H curve.

Important **For more information about materials in the database, contact the manufacturer of each material.**

2-7-2. Material Database Table

The materials contained in the JMAG and their manufacturers are listed in the following table.

Manufacturer	JMAG-Studio	JMAG-Designer
Hitachi	HS	Sintered NdFeB
Hitachi Metal (SSMC)	Rare earth	
Hitachi Metals	-	
JSOL	-	
Shin-Etsu	Nseries	
TDK	Sintered NdFeB	
DEC	Radially Oriented Ring	Bonded NdFeB
	Injection Molded	
	Compression Molded	
Hitachi	HB	
Hitachi Metal (SSMC)	Bonded	
Hitachi Metals	-	
MSM	Nd Bonded Magnet	
Tokyo Ferrite MFG	NdFeB Compression	
	NdFeB Injection	
Hitachi	Ferrite magnet	Ferrite
Hitachi Metal (SSMC)		
Hitachi Metals		
JSOL		
TDK		
Tokyo Ferrite MFG	Ferrite Rubber	
	Ferrite Injection	
	Ferrite Sinter	
	SmFeN Injection	SmFeN
Shin-Etsu	Rseries	SmCo
TDK	Sm-Co	
MSM	FeCrCo Magnet	FeCrCo
	Vicalloy Magnet	FeCoV
Hitachi Metal (SSMC)	Alnico	AlNiCo
MSM	MK Magnet	

Manufacturer	JMAG-Studio	JMAG-Designer
China Steel	-	Steel Sheet
JFE Steel	JN Core	
	JNA Core	
JSOL	-	
Nippon Steel	Non-oriented	
SMI	Semi-processed Type	
	Fully-processed Type	
VAC	Steel Sheet	
DaidoSteel	Adjustalloy-L (Fe-Si)	Soft Magnetic Composite
	Adjustalloy-L (Fe-Si-Al)	
HoganasAB	-	
Hitachi Metals	-	Soft Ferrite
JFE Ferrite	Ferrite	
JSOL	Mn-Zn	
	Ni-Zn	
TDK	-	
Hitachi Metals	Permalloy	Permalloy
JSOL		
VAC		
Hitachi	Permendur	Permendur
MSM		
VAC		
JSOL	Steel for Structure	Steel for Structure
	-	Linear Material

MEMO The database of Hitachi contains the data for demagnetization.

2-7-3. China Steel

■ Material

- Steel Sheet

■ Important Notes Regarding the Use of This Function

- Tests were conducted in accordance with JIS C 2550(2000) method.
- The data is one example of sampling and for reference only.

■ Contact

China Steel Corporation (Taiwan)

TECHNICAL SERVICE SECTION
METALLURGICAL DEPARTMENT
Address: 1 Chung Kang Road Hsiao Kang, Kaohsiung 81233 Taiwan, R.O.C
Phone: +886-7-802-1111 ext 5378
Fax: +886-7-803-9553

2-7-4. DAIDO ELECTRONICS

■ Material

- Bonded NdFeB

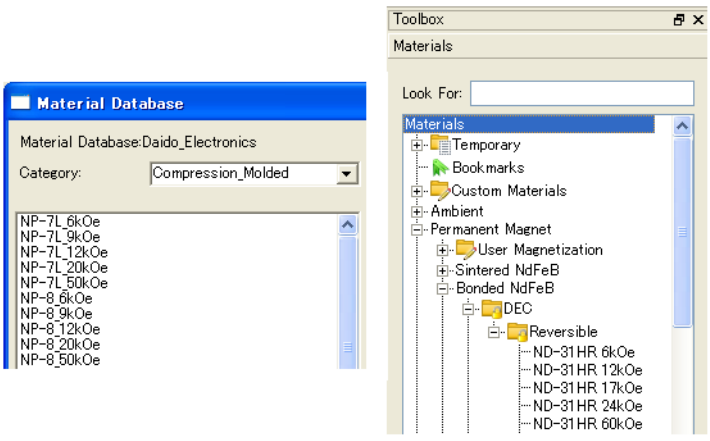
■ Important Notes Regarding the Use of This Function

- Measurement conditions are as follows.
  - Temperature: room temperature
  - Sample: ϕ107, Pc=2.
  - Coating: None
- As Noquench-DR is a radial anisotropic ring magnet, it has the magnetization properties in the radial direction of the magnet.

- The density of a magnet varies as shown in the table below. JMAG uses typical values.

		Data in the JMAG		
		Electric	Density	Density range
		μohm*m	kg/m3	kg/m3
Injection molded	NPI-4L	130	4550	4200-4900
	NPI-5L	130	5150	4900-5400
	NPI-6	130	5250	5000-5500
	NPI-6L	130	5250	5000-5500
	NPI-8L	130	5250	5000-5500
	NPI-8LL	130	5250	5000-5500
	NPI-6LR	130	4850	4600-5100
	NPI-6SR	130	4850	4600-5100
Compression molded	NP-7L	56	5850	5600-6100
	NP-8	56	5850	5600-6100
	NP-8L	56	5850	5600-6100
	NP-8R	56	5850	5600-6100
	NP-8SR	31	5850	5600-6100
	NP-10LL	56	6050	5900-6200
	NP-11L	56	6050	5900-6200
	NP-12L	56	6250	6100-6400
MQ3	ND-31HR	1.35	7600	
	ND-31SHR	1.35	7700	
	ND-35HR	1.35	7600	
	ND-39R	1.35	7600	

MEMO Each material in the table is further divided into categories depending on the strength of the magnetization field. The suffix "(number)kOe" of the material name indicates the strength of the magnetization field.



JMAG-Studio JMAG-Designer

■ Contact

DAIDO ELECTRONICS Co., LTD

- Tokyo Sales Office

  Person of charge: Aizaki

  Address: Daido Shinagawa Building, 1-6-35, Minato-Minami, Minato-ku, Tokyo 108-0075, Japan
  Phone: +81-3-5495-1851

- Nagoya Sales Office

  Person of charge: Haizuka

  Address: 21F, Urbannet Building, 1-1-10, Higashisakura, Higashi-ku, Nagoya 461-0005, Japan
  Phone: +81-52-963-7560

2-7-5. Daido Steel

■ Material

- Soft Magnetic Composite

■ Important Notes Regarding the Use of This Function

- The data represents typical values.
- Electrical resistivity is variable over a rather broad range depending on the amount of binder, and a typical value is shown.

■ Contact

Daido Steel Co., Ltd

Research & Development Lab.
Electromagnetic Research Dept.
Magnetic Materials Research Team

Phone: +81-52-611-9424

2-7-6. Hoganas AB

■ Material

- Soft Magnetic Composite

■ Important Notes Regarding the Use of This Function

- Compaction

  0.5% Kenolube and 0.6% LB1 CC are the values at room temperature.

  The data of 0.6% LB1 WC was obtained by warm compaction.
- The heat treatment conditions is recommend to use.

  The following conditions are based on the components' temperatures.

  0.5% Kenolube: Heated at 500 degrees Celsius for 15 minutes.

  0.6% LB1: Cured at 225 degrees Celsius for 60 minutes.
- Density after heat treatment/curing

  0.5 % Kenolube: 6.87 g/c· (400MPa), 7.18 g/c· (600MPa), 7.35g/c·(800 MPa)

  0.6 % LB1 CC: 6.83 g/c· (400MPa), 7.19 g/c· (600MPa), 7.29 g/c· (800 MPa)

  0.6 % LB1 WC: 7.02 g/c· (400MPa), 7.24 g/c· (600MPa), 7.31 g/c· (800 MPa)

■ Contact

Hoganas AB

MM2

The Person of Charge: Mats Persson

Address: S-263-83 Hoganas, Sweden

Phone: +46-42-33-8307

Fax: +46-42-33-8150

2-7-7. Hitachi Metals

■ Materials

- Sintered NdFeB
- Bonded NdFeB
- Ferrite
- AlNiCo
- Soft Ferrite
- Permalloy
- Permendur

■ Important Notes Regarding the Use of This Function

- The data represents the average values.
- Nd-Fe-B rare-earth magnet of the radial anisotropic type has the following measurement constraints.

  Inside diameter (Di)/outside diameter (Do)=0.75~0.95

  Outside diameter (Do)=10 mm~150 mm
- The properties may not be available or applicable to a magnet depending on its shape and measurements.

MEMO Sumitomo Special Metals Co., Ltd. has changed their corporate name to NEOMAX Co.,Ltd. as of April 1, 2004. Effective April 1, 2007,Hitachi Metals, Ltd. and NEOMAX merged with Hitachi Metals as the surviving corporation and NEOMAX as the dissolved corporation.

■ Contact

Hitachi Metals, Ltd. NEOMAX Company

- Tokyo Sales Division

  Address: SEAVANS North Bldg. 2-1, Shibaura 1-chome, Minato-ku Tokyo 105-8614, Japan
  Phone: +81-3-5765-4202
  Fax: +81-3-5765-4457
- Chubu-Tokai Sales Office

  Address: Takisada Building, 2-13-19, Nishiki, Naka-ku, Nagoya-shi, Aichi-ken 460-0003, Japan
  Phone: +81-52-220-7469
  Fax: +81-52-201-5188

- Kansai Sales Office

  Address: Nissei Yodoya-bashi Building, 3-5-29, Kitahama, Chuo-ku, Osaka-shi, Osaka-fu 541-0041, Japan
  Phone: +81-6-6203-9702
  Fax: +81-6-6222-3417
- Kyushu Sales Office

  Address: Taisei Hakata Station East Building, 1-9-11, Hakata Station East, Hakata-ku, Fukuoka-shi, Fukuoka-ken 812-0013, Japan
  Phone: +81-92-432-8614
  Fax: +81-92-432-8620

2-7-8. JFE Ferrite

■ Material

- MnZn Ferrite

■ Important Notes Regarding the Use of This Function

- The data presents the average values shown in the catalog. The values were measured with the test core (R31/19/8A). In actual product cores, please consider that there is the characteristic dispersion.

■ Contact

JFE Ferrite Corporation

Sales Department
Address: JFE Kuramae Building 6F, 2-17-4, Kuramae, Taito-Ku, Tokyo 111-0051, Japan
Phone: +81-3-3863-7951
Fax: +81-3-3863-7112

JFE CHEMICAL CORPORATION

Magnetic Materials R&D Center, Chemical Research Laboratory
Address: 1 Mizushima Kawasaki-dori, Kurashiki, Okayama 712-8074, Japan
c/o JFE Ferrite Corp.
Phone: +81-86-447-4212
Fax: +81-86-446-2154

2-7-9. JFE Steel

■ Material

- Steel Sheet

■ Important Notes Regarding the Use of This Function

- The values were measured using the method JIS C 2550 (1992) with the test piece consisting of half the strip sheared parallel and the other half sheared transverse to the rolling direction.
- The data represents typical values and not the guaranteed values.

  JN Core: The iron loss and flux density were measured with the shear.

  JNA Core: The iron loss and flux density are the values after stress relief annealing at 750 degrees Celsius (1380 Fahrenheit), *2Hrs. under neutral atmosphere.
- Flux density B is the value defined by following equations.

  $B=J+\mu_0 H$

  J: Magnetic polarization μ_0: Permeability in vacuum = 4π*10-7 H/m

■ Contact

JFE STEEL CORPORATION

Steel Research laboratory

Address: 1 Mizushima Kawasaki-dori, Kurashiki, Okayama 712-8511, Japan

Phone: +81-86-447-3959

Fax: +81-86-447-3951

2-7-10. MITSUBISHI STEEL

■ Materials

- Bonded NdFeB
- FeCrCo
- FeCoBa
- AlNiCo
- Permendur

■ Important Notes Regarding the Use of This Function

- The data represents the standard values at room temperature in the catalog.
- The properties of Vicalloy is of when used as a permanent magnet.
- The properties of a semi-hard magnetic material vary depending on the heat treatment.

■ Contact

MITSUBISHI STEEL MFG. CO., LTD.

Engineering and Development Group Utsunomiya Works
Address: 1, Hiraide Kogyodanchi,Utsunomiya 321-0905, Japan
Phone: +81-28-661-4929
Fax: +81-28-664-1685

2-7-11. Nippon Steel

■ Material

- Steel Sheet

■ Important Notes Regarding the Use of This Function

- The data represents typical values.

■ Contact

Nippon Steel & Sumitomo Metal Corporation

Phone: +81-3-6867-6619

2-7-12. Shin-Etsu Chemical

■ Materials

- Sintered NdFeB
- SmCo

■ Important Notes Regarding the Use of This Function

- The data is the average values shown in the catalog and not guaranteed values.
- The electric resistivity was measured at room temperature.

■ Contact

Shin-Etsu Chemical Co., Ltd.

Electric Materials Division Magnet dept.

Phone: +81-3-3246-5246

Fax: +81-3-3246-5367

2-7-13. Sumitomo Metal Industries

■ Material

- Steel Sheet

■ Important Notes Regarding the Use of This Function

- Full-processed type:
  All the test pieces are measured in accordance with JIS C 2550(2000), “Electrical Steel Sheets”.
  Note: Magnetic induction $B = J + \mu_0 H$
- Semi-processed type:
  All the test pieces are annealed in N2 gas at 750 Celsius for 2 hours.
  All the test pieces are measured in accordance with JIS C 2550(2000), “Electrical Steel Sheets”.
  Note: Magnetic induction $B = J + \mu_0 H$

■ Contact

Nippon Steel & Sumitomo Metal Corporation

Phone: +81-3-6867-6619

2-7-14. TDK

■ Materials

- Sintered NdFeB
- Ferrite
- SmCo
- Soft Ferrite

■ Important Notes Regarding the Use of This Function

- The magnet database (TDK) offers the typical properties of each material to be simulated. Please note that there may be difference between the data and the actual products due to variations.

■ Contact

TDK Corporation

Magnetic Product Business Group

Address: 1-13-1 Nihonbashi, Chuo-ku, Tokyo 103-8272, Japan
Phone: +81-3-5201-7279
Fax: +81-3-5201-7169

2-7-15. Tokyo Ferrite

■ Materials

- Bonded NdFeB
- Ferrite
- SmFeN

■ Important Notes Regarding the Use of This Function

- Measurement values are typical.
- Measurement temperature is 23 Celsius ± Celsius.

■ Contact

Tokyo Ferrite MFG Co., Ltd.

Sales Department

The Person of Charge: Hajime Uda

Mail: h.uda@tokyoferrite-ho.co.jo

Address: 1-14-1 Tabatashinmachi, Kita-ku, Tokyo 114-8528, Japan

Phone: +81-3-3893-9501

Fax: +81-3-3810-4661

2-7-16. VAC

■ Materials

- 80% NiFe (Permalloy)
- 50% NiFe
- 50% CoFe (Permendur)
- 17-18% CoFe
- Reference 3%SiFe Electrical steel

■ Important Notes Regarding the Use of This Function

- The data represents typical values.

■ Contact

VACUUMSCHMELZE GmbH & CO.KG

Mail: info@vacuumschmelze.com

Address: Gruener Weg 37, D-63450 Hanau, Germany

Phone: +49-6181-380

2-7-17. JSOL (Sample Material Data)

■ Materials

- Bonded Needful
- Ferrite
- Steel Sheet
- Proemial
- Linear Material
- Structural Material

■ Important Notes Regarding the Use of This Function

- The following should be noted when using data other than structural material.
  - The data is not measured values but was created based on a function of magnetizing properties.

    Example 1: permalloy_150000
    - Initial magnetic permeability: μ_i=150000
    - Saturated magnetic flux density: B_5=0.65

    Example 2: Mn-Zn_ferrite_1200
    - Initial Magnetic Permeability: μ_i=1200
    - Saturated Magnetic Flux Density: B_5=0.55
- The following should be noted when using structural material.
  - The value needs to be an example value and not a guaranteed value.
  - Measured value of ambient temperature.
  - Test piece for S45C, S10C, and STKM16A in atmosphere 900 Celsius x implementing 1 hour of air cooling.

2-8. Coordinate System

Use the local coordinate system or cylindrical coordinate system to define the lamination direction or the easy-axis direction of an anisotropic material.

The following should be noted when using this function.

- If a subroutine *magusr* (345 page) is used, flux density vector B' on the local coordinate system will be assigned to the subroutine.

▶▶ Reference

- For more information, see “Working with Coordinate System” in Chapter 3 Basic Operation of JMAG-Studio in the JMAG-Studio User's Manual.
- For more information, see Chapter 37 “Coordinate System” in the JMAG-Designer User's Manual.

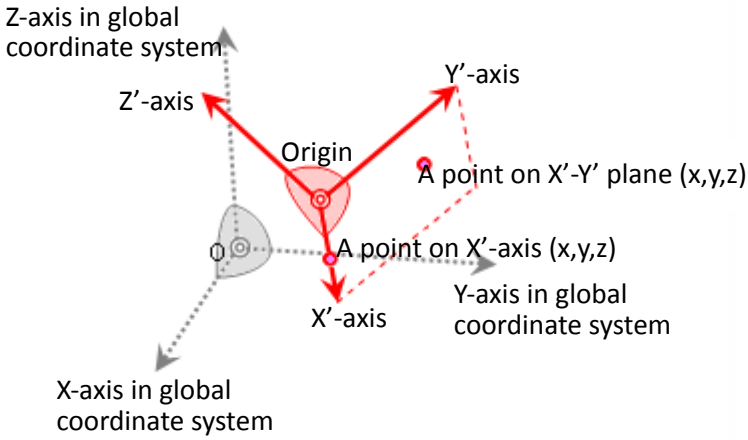
2-8-1. Rectangular Coordinate System

- The rectangular coordinate system consists of the X-direction, Y-direction, and Z-direction.
- In JMAG-Studio, the local coordinate system can be defined in the following ways.
  - Specify 3 points
  - Specify 2 sides and 1 point
  - Specify 1 side and 2 points
  - Specify 1 plane, 1 side, and 1 point

For instance, if selecting the specify 3 points method to match the local coordinate system with the global coordinate system, use the following settings.

- Coordinates of the origin in the global coordinate system: (0,0,0)
- Coordinates of a point on the X-axis in the global coordinate system: (1,0,0)

- Coordinates of a point on the XY plane in the global system: (0,1,0)

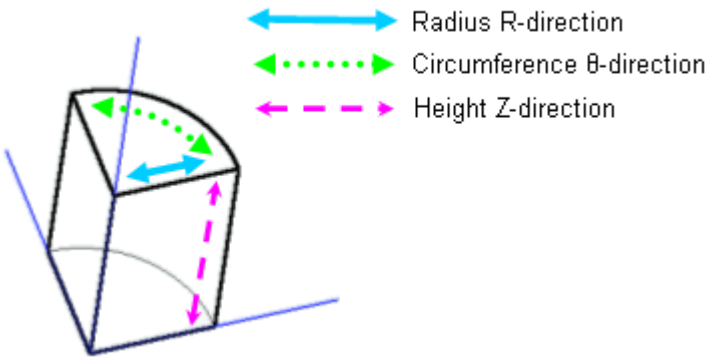


Local coordinate system (defined by 3-point specified)

- In JMAG-Studio, the defined local coordinate system can be translated and rotated.
  - To translate, define the X-direction, Y-direction, and Z-direction.
  - To rotate, define the X-axis, Y-axis, and Z-axis.

2-8-2. Cylindrical Coordinate System

- The cylindrical coordinate system consists of the radial direction, circumferential direction, and height direction.



Cylindrical coordinate system

2-9. Material Types of Magnetic Field Analysis (JMAG-Studio)

The type of material needs to be selected before creating the material properties in JMAG-Studio. The properties that can be set are different for each type of material. The proper type needs to be selected.

■ Air

- The air is treated as a material having the relative permeability of 1.0 and the electrical conductivity of 0.0 (1/(ohm・m)).
- The gap element can be set.

■ Coil

- The coil material has the relative permeability of 1.0 and the electric conductivity of 0.0 (1/(ohm・m)).
- [Coil] needs to be selected as the type of material if an FEM coil condition or current condition is to be set.

MEMO In the following cases, select [Non Magnetic Material] instead of [Coil], and then specify the electric conductivity.
- When eddy currents are distributed inside the coils
- When Joule losses are obtained by setting a current condition.

■ Non-magnetic material

- The non-magnetizing material has the relative permeability of 1.0.
- The electric properties can be set.
- The gap element (68 page) can be set.

■ Isotropic magnetic material

- The magnetizing properties can be set.
  - The magnetizing properties can be selected from the material database (44 page).
  - The magnetizing properties can be specified with a point sequence.
  - The stress-dependent magnetizing properties can be set.
  - The magnetizing properties can be corrected.
- The electric properties can be set.
- The gap element (68 page) can be set.

■ Isotropic magnetic steel sheet

- The magnetizing properties can be set.
  - The magnetizing properties can be selected from the material database (44 page).
  - The magnetizing properties can be specified with a point sequence.
  - The stress-dependent magnetizing properties can be set.
  - The magnetizing properties can be corrected.
  - The lamination direction and lamination factor can be set.
- The electric properties can be set.
  - The properties can be set in the in-plane direction and lamination direction.

■ Anisotropic magnetic steel sheet

- The magnetizing properties can be set.
  - The magnetizing properties can be selected from the material database (44 page).
  - The magnetizing properties can be specified with a point sequence.
  - The magnetizing properties can be corrected.
  - The anisotropy/lamination direction can be set.
  - Different magnetizing properties can be set for the easy axis direction and the hard axis direction.
  - The lamination direction and lamination factor can be set.
- The electric properties can be set.
  - The properties can be set in the in-plane direction and lamination direction.

■ Magnet

- The demagnetization properties can be set using one of the following ways.
  - Select from the material database (44 page).
  - Specify the B-H curve
  - Specify the coercive force and relative permeability.
- The magnetization direction can be set using one of the following ways.
  - Select a pattern.
  - Use the magnetization tool (program). If the magnetization tool is used, the demagnetization distribution can be obtained because the magnetization ratio is taken into account.
- The magnetizing properties can be corrected.
- The electric properties can be set.
- The temperature-dependent magnetizing properties can be set.

Important **Magnets cannot be used for frequency response analysis (FQ).**

2-10. Superconductivity Properties (JMAG-Studio)

In JMAG-Studio, the superconductivity properties can be set for a 3D transient response analysis. The successive iteration method is used for nonlinear iterative calculation when the superconductivity properties are set.

2-10-1. Superconductor Model Types

- There are three types of superconductor: Bean model, I-V model and user-defined model. The properties of each superconductor are defined by specifying the following parameters:

■ Bean model

- Initial electrical conductivity (1/(ohm・m))
- Maximum electrical conductivity (1/(ohm・m))
- Critical current density (A/m^2)

The Beam model is defined as follows:

If $|J| > J_c$, then $\sigma_n = \frac{J_c}{E_n}$

If $|J| < J_c$, then $\sigma_n = \sigma_{init}$

where J, σ_n, E_n, J_c and $\sigma\,init$ represent the current density, electrical conductivity, electric field, critical current density and initial electrical conductivity, respectively.

If the current density does not reach its critical value for each element, set a greater value for the electrical conductivity so that the critical current flows. Conversely, if the current density exceeds its critical value, set a smaller value for the electrical conductivity.

Perform the above change during the nonlinear iterative calculation and repeat the process until the rate of the electrical conductivity changes falls below a certain value. To judge the convergence of electrical conductivity, use the value of Iterative Convergence Tolerance. The default value is 0.001.

For a Bean model, the upper limit of the electrical conductivity is only a reference, and is not used in the calculation process. Set the same value as the initial electrical conductivity for the maximum electrical conductivity.

■ I-V model

- Initial electrical conductivity (1/(ohm・m))
- Maximum electrical conductivity (1/(ohm・m))
- Critical current density (A/m^2)
- *n* (1 or larger real number)

The I-V model is defined as follows:

$$E = E_c\left(\frac{J}{J_c}\right)^n$$

where J, E, J_c and E_c represent the electric current density, electric field, critical current density and critical electric field, respectively. As with the Bean model, change the electrical conductivity according to the current density at each element.

The difference from the Bean model is that the value *n* is used in the equation. The value *n* is a real number of 1 or larger. In the I-V model, the properties are identical to a normal electric conductor when *n=1*, and it approaches the Bean model as n increases. The critical electric field is defined by the following equation.

$$E_c = \frac{J_c}{\sigma_{init}}$$

Note that the initial electrical conductivity determines the critical electric field.

■ User component model

- Initial electrical conductivity (1/(ohm・m))
- Maximum electrical conductivity (1/(ohm・m))

Describe the superconductor properties using a user subroutine *usrsc* (385 page).

The initial electrical conductivity is used for the first nonlinear iterative calculation. The subroutine is called from JMAG in the second or later nonlinear iterations. The electrical conductivity value temporarily used in the nonlinear iterations is calculated by JMAG based on the electrical field and current density defined by the subroutine. The upper limit of this value is determined by the maximum electrical conductivity of superconductor.

2-10-2. Critical Current Density

- Magnetic flux dependence can be specified for the critical current density. To do the above, select the [Magnetic Flux Density Dependence] radio button from the [Critical Current Density] group box in the [Superconductor] dialog box, and specify the magnetic flux density of the critical current density using a point sequence.
- If using the direction component of magnetic flux density, specify the unit direction vector. In that case, the critical current density is determined by the dependent point sequence based on the projection components in the unit direction specified by the flux density vectors for each element.
- If using the absolute magnetic flux density value, the critical current density is determined by the dependent point sequence based on the absolute value of the magnetic flux density.

2-10-3. Setting Anisotropy for the Electrical Conductivity of Superconductors

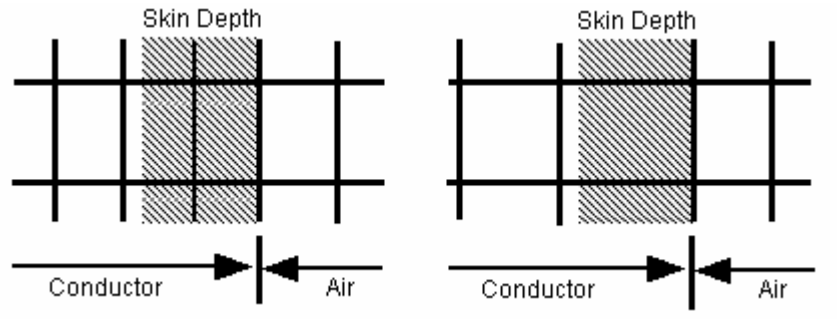
- Anisotropy can be specified for the electrical conductivity. To do the above, select the [Anisotropic] radio button from the [Isotropy/Anisotropy] group box in the [Superconductor] dialog box.
- Anisotropy is defined using a user subroutine *conusr* (349 page). The user subroutine is called by JMAG in each nonlinear iteration for each element. Anisotropy can be represented by modifying the electrical conductivity tensor *ij* in the user subroutine *consur*f.

  The electrical conductivity tensor σij must be symmetrical. The electric properties of superconductors are represented by using a temporary electrical conductivity which is assigned to the diagonal component of tensor σ passed by JMAG. Initially, after it is passed by JMAG, tensor σ has off diagonal components of zero. So, modify tensor σ according to the anisotropy to be set.

2-10-4. Notes on Subdividing the Elements

- Set the element size in the superconductor region to be the same as or half the skin depth which is obtained by the upper limit of electrical conductivity, relative permeability of the conductor, and the frequency at which a current is generated.

  If the element is too small, discontinuity specific to the superconductivity may not be expressed. On the contrary, if it is too large, the calculation accuracy may be reduced.



Good Example Poor Example

Relationship of skin depth and mesh size

2-11. Gap Element (JMAG-Studio)

The areas where a magnetic field penetrates the elements perpendicularly, such as a gap in the motor can be represented by the shell elements that have no thickness, instead of solid elements that have a thickness.

In JMAG-Studio, the gap elements can be set for each material.

- This feature is only available for 3D analysis.
- This feature assumes that the magnetic field penetrates the gap element perpendicularly. If it is not the case, there may be some error.
- This feature supports only shell elements and is not available for solid elements.
- The following conditions cannot be set for gap elements.
  - Symmetry boundary
  - Current
  - Current density
  - FEM coil
  - FEM conductor
  - Slide condition
- Do not create gap elements overlapping each other on the same plane. The gap elements may not be taken into account during analysis.

The gap elements can be set as follows.

1. **Create shell elements to set the gap elements.**
2. **Set the material properties of the shell elements.**
   In the [others] tab in the [Material] dialog box, select the [Gap Element] dialog box, and then enter the thickness of the gap elements.

▶▶ Reference

The gap elements can be set using the gap condition. For details, see P.262 "3-33. Gap".

Chapter 3. Function of Magnetic Field Analysis

This chapter describes the material properties and conditions used in the magnetic field analysis.

▶▶ Reference

For information about setting material properties and conditions, see the JMAG-Studio User's Manual or JMAG-Designer User's Manual.

3-1. Overview of Functions

The following modules are available for the magnetic field analysis.

- Static analysis
  - 3D magnetic field analysis (ST 3D)
  - 2D magnetic field analysis (ST2D)
  - Axial symmetry magnetic field analysis (STAX)
- Transient response analysis
  - 3D magnetic field analysis (TR3D)
  - 2D magnetic field analysis (DP2D)
  - Axial symmetry magnetic field analysis (DPAX)
- Frequency response analysis
  - 3D magnetic field analysis (FQ3D)
  - 2D magnetic field analysis (FQ2D)
  - Axial symmetry magnetic field analysis (FQAX)

The following module is also available for the magnetic field analysis.

- Transformer analysis

This chapter describes each module and the features of each analysis.

Important **The transformer analysis is only supported by JMAG-Designer.**

▶▶ Reference

- For information about the license types and analysis types available for JMAG-Studio, see Chapter 2 “About JMAG-Studio” in the JMAG-Studio User's Manual.

- For information about the license types and analysis types available for JMAG-Designer, see Chapter 2 “Features of JMAG-Designer” and Chapter 26 “Creating a Study by Selecting an Analysis Type” in the JMAG-Studio User's Manual.
- For information about the iron loss analysis, P.872 "12-3. Calculate Iron Loss".

3-1-1. ST (Magnetic Field Analysis (Static Analysis) Module)

- The static analysis is used when the magnetomotive force such as current and the position of objects do not vary with time.
- Linear and nonlinear materials can be used.
- The FEM that applies the vector potential on the element edge as an unknown value is used for a 3D analysis.
- The FEM that applies the vector potential on a node as an unknown value for the conductors carrying eddy currents and magnetic materials is used for a 2D or axisymmetric analysis.
  The FEM+BEM can be used for the non-magnetic materials such as air. In this case, elements need not to be generated in the air region.
- The Y-axis must be the center axis for an axisymmetric analysis.

▶▶ Reference

For information about linear and nonlinear materials, see P.78 "3-4. Material Property for Each Part: Permanent Magnet".

3-1-2. TR/DP (Magnetic Field Analysis (Transient Response Analysis) Module)

- The transient response analysis is used when the magnetomotive force such as current and the position of objects vary with time.
- Linear and nonlinear materials can be used.
- If the transient response analysis is used with thermal analysis (HT), the temperature-dependent magnetizing properties and electric properties can be included in the analysis (coupled analysis).
- The FEM that applies the vector potential on the element edge as an unknown value is used for a 3D analysis.

- The FEM that applies the vector potential on a node as an unknown value for the conductors carrying eddy currents and magnetic materials for a 2D or axisymmetric analysis.
  The FEM+BEM can be used for the non-magnetic materials such as air. In this case, elements need not to be generated in the air region.

MEMO The Y-axis must be the center axis for an axisymmetric analysis.

▶▶ Reference

- For information about linear and nonlinear materials, see P.78 "3-4. Material Property for Each Part: Permanent Magnet".
- For information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

3-1-3. FQ (Magnetic Field Analysis (Frequency Response Analysis) Module)

- The frequency response analysis is used when the magnetomotive force such as current varies sinusoidally with time, and is run for a single frequency.
- Linear and nonlinear materials can be used. In a nonlinear analysis, the actual value of flux density is used to run iterative calculations of the nonlinear magnetizing property curve. Therefore, the error may be larger when the material has a strong nonlinearity.
- A permanent magnet cannot be used.
- A physical quantity that varies sinusoidally with time is output in a complex number.
- If the frequency response analysis is used with thermal analysis (HT), the temperature-dependent magnetizing properties and electric properties can be included in the analysis (coupled analysis).
- The FEM that applies the vector potential on the element edge as an unknown value is used for a 3D analysis.
- The FEM that applies the vector potential on a node as an unknown value for the conductors carrying eddy current and magnetic materials is used for a 2D or axisymmetric analysis.
  The FEM+BEM can be used for the non-magnetic materials such as air. In this case, elements need not to be generated in the air region.
- The Y-axis must be the center axis for an axisymmetric analysis.

▶▶ Reference

- For information about linear and nonlinear materials, see P.78 "3-4. Material Property for Each Part: Permanent Magnet".
- For information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

3-1-4. TS (Transformer Analysis Module) (JMAG-Designer)

- The transformer analysis is used to run a specialized electromagnetic field analysis for transformers, reactors, inductors, etc.
- The magnetic field analysis (static analysis, transient response analysis, and frequency response analysis) is used for a transformer analysis.
- The results of a transformer analysis can be used for a iron loss analysis, thermal analysis, and structural analysis (one way coupled analysis).

▶▶ Reference

- For information about the transformer analysis, see the JMAG-Designer User's Manual.
- For information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

MEMO For JMAG-Studio, 3D rotation machine analysis and 2D rotation machine analysis are available. The 3D rotation machine analysis is the same as the 3D transient response magnetic-field analysis. The 2D rotation machine analysis is the same as the 2D transient response magnetic field analysis.

MEMO For JMAG-Designer, section transient response analysis (2D) and section transient response analysis (axisymmetric) are available.

3-2. Conditions of Magnetic Field Analysis (JMAG-Designer)

The analysis conditions supported in JMAG are listed below. Some conditions may not be available depending on the module used.

Analysis conditions	Module*1								
	ST3D	ST2D	STAX	TR3D	DP2D	DPAX	FQ3D	FQ2D	FQAX
Material properties									
Magnetic properties	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙
Electric properties	X	X	X	O	O	O	O	O	O
Magnet database	O	O	O	O	O	O	X	X	X
Magnetic steel sheet database	O	O	O	O	O	O	O	O	O
Permanent magnet	O	O	O	O	O	O	X	X	X
Boundary									
Rotation periodic boundary	O	O	X	O	O	X	O	O	X
Translation periodic boundary	O	O	O	O	O	O	O	O	O
Symmetry boundary	⊙	O	O	⊙	O	O	⊙	O	O
Natural boundary	O	O	O	O	O	O	O	O	O
Gap flux boundary	X	X	X	X	X	X	O	X	X
Slide *3	X	X	X	O	O	O	O	O	O
Current									
Current	O	O	O	O	O	O	O	O	O
Current density	O	O	O	O	O	O	O	O	O
FEM coil *2	X	X	X	O	O	O	O	O	O
Group FEM coil	X	X	X	O	O	X	O	O	X
FEM conductor *2	X	X	X	O	O	O	O	O	O
Group FEM conductor	X	X	X	O	O	X	O	O	X
Motion									
Rotation motion	X	X	X	O	O	X	O	O	X
Translation motion	X	X	X	O	O	O	O	O	O

Analysis conditions	Module*1								
	ST3D	ST2D	STAX	TR3D	DP2D	DPAX	FQ3D	FQ2D	FQAX
Output									
Force (nodal force, Lorentz force, surface force)	O	O	O	O	O	O	O	O	O
Torque (nodal force, Lorentz force, surface force)	O	O	X	O	O	X	O	O	X
Heat source	X	X	X	O	O	O	O	O	O
Iron loss	O	O	O	O	O	O	O	O	O
Magnetic flux	O	O	O	O	O	O	O	O	O
Magnetostriction	O	O	O	O	O	O	O	O	O
Partial Inductance	X	X	X	X	X	X	O	X	X
Link									
Partial Model	O	O	O	O	O	O	O	O	O
External circuit link 1/2	X	X	X	O	O	O	X	X	X
Subcycling	X	X	X	O	O	O	X	X	X
Frozen permeability	O	O	O	O	O	O	O	O	O
Stress distribution	O	O	O	O	O	O	O	O	O
Temperature distribution	O	O	O	O	O	O	O	O	O
Displacement	O	O	O	O	O	O	O	O	O
Reusing Demagnetization	O	O	O	O	O	O	X	X	X
Modeling									
External field	O	X	X	O	X	X	O	X	X
Gap	O	X	X	O	X	X	O	X	X
Insulation	X	X	X	O	O	X	O	O	X
Circuit									
External circuit	X	X	X	O	O	O	O	O	O
Study properties									
Calculation method	O	O	O	O	O	O	X	X	X
Step control	⊙	⊙	⊙	⊙	⊙	⊙	X	X	X
Frequency control	X	X	X	X	X	X	⊙	⊙	⊙
Coupled analysis	O	O	O	O	O	O	O	O	O
Full model conversion	O	O	O	O	O	O	O	O	O
Circuit conversion	X	X	X	O	O	O	O	O	O
ICCG	O	O	O	O	O	O	O	O	O

Analysis conditions	Module*1								
	ST3D	ST2D	STAX	TR3D	DP2D	DPAX	FQ3D	FQ2D	FQAX
Nonlinear calculation	O	O	O	O	O	O	O	O	O
Output control	O	O	O	O	O	O	O	O	O
Snapshot calculation	X	X	X	O	O	O	O	O	O
Restart	X	X	X	O	O	O	O	O	O
User subroutine	O	O	O	O	O	O	O	O	O

⊙: Required O: Optional X: Not available

*1 Module names and corresponding analysis types are as follows:

- ST3D: 3D magnetic field analysis (static analysis)
- ST2D: 2D magnetic field analysis (static analysis)
- STAX: Axial symmetry magnetic field analysis (static analysis)
- FQ3D: 3D magnetic field analysis (frequency response analysis)
- FQ2D: 2D magnetic field analysis (frequency response analysis)
- FQAX: Axial symmetry magnetic field analysis (frequency response analysis)
- TR3D: 3D magnetic field analysis (transient response analysis)
- DP2D: 2D magnetic field analysis (transient response analysis)
- DPAX: Axial symmetry magnetic field analysis (transient response analysis)

*2 The FEM coil condition and FEM conductor condition cannot be used independently. They can only be used for an analysis with a circuit.

*3 A slide condition must always be set with a motion condition.

▶▶ Reference

- For information about the conditions that are available in the section analysis, see Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.
- For information about the conditions that are available in the transformer analysis, see Chapter 33 “Study Properties and Conditions of Trans Analysis” in the JMAG-Designer User's Manual.

3-3. Conditions of Magnetic Field Analysis (JMAG-Studio)

The analysis conditions supported in JMAG are listed below. Some conditions may not be available depending on the module used.

Analysis conditions	Module*1								
	ST3D	ST2D	STAX	TR3D	DP2D	DPAX	FQ3D	FQ2D	FQAX
Material properties									
Magnetizing properties	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙
Magnet database	O	O	O	O	O	O	X	X	X
Magnetic steel sheet database	O	O	O	O	O	O	O	O	O
Permanent magnet (magnetization direction)	O	O	O	O	O	O	X	X	X
Electric properties	X	X	X	O	O	O	O	O	O
Superconductivity properties	X	X	X	O	X	X	X	X	X
Conditions									
Step	⊙	⊙	⊙	⊙	⊙	⊙	X	X	X
Frequency	X	X	X	X	X	X	⊙	⊙	⊙
Symmetry boundary 2D	X	O	O	X	O	O	X	O	O
Symmetry boundary 3D	⊙	X	X	⊙	X	X	⊙	X	X
Periodic boundary	O	O	X	O	O	X	O	O	X
Current	O	O	O	O	O	O	O	O	O
Current density	O	O	O	O	O	O	O	O	O
External circuit	X	X	X	O	O	O	O	O	O
Coupling with external circuit simulator	X	X	X	O	O	O	X	X	X
Subcycling	X	X	X	O	O	O	X	X	X
FEM coil *2	X	X	X	O	O	O	O	O	O
Group FEM coil	X	X	X	O	O	X	O	O	X
FEM conductor *2	X	X	X	O	O	O	O	O	O
Group FEM conductor	X	X	X	O	O	X	O	O	X
Electromagnetic force / torque calculation	O	O	O	O	O	O	O	O	O
Magnetic flux calculation	O	O	O	O	O	O	O	O	O
Motion	X	X	X	O	O	O	O	O	O
Slide *3	X	X	X	O	O	O	O	O	O
External field	O	X	X	O	X	X	O	X	X
Insulation	X	X	X	O	X	X	O	X	X

Analysis conditions	Module*1								
	ST3D	ST2D	STAX	TR3D	DP2D	DPAX	FQ3D	FQ2D	FQAX
Gap	O	X	X	O	X	X	O	X	X
Stress distribution	O	O	O	O	O	O	O	O	O
Iron loss	O	O	O	O	O	O	O	O	O
Partial inductance	X	X	X	X	X	X	O	X	X
Adaptive mesh control	O	O	O	O	O	O	O	O	O
Heat Source	X	X	X	O	O	O	O	O	O
Partial Model	O	O	O	O	O	O	O	O	O
Analysis control									
Basic setting	O	O	O	O	O	O	O	O	O
Restart	X	X	X	O	O	O	O	O	O
Snapshot analysis	X	X	X	O	O	O	O	O	O
Circuit conversion	X	X	X	O	O	O	O	O	O
ICCG method	O	O	O	O	O	O	O	O	O
Nonlinear calculation	O	O	O	O	O	O	O	O	O
Post output	O	O	O	O	O	O	O	O	O
User subroutine	O	O	O	O	O	O	O	O	O

⊙: Required O: Optional X: Not available

*1 Module names and corresponding analysis types are as follows:

- ST3D: 3D magnetic field analysis (static analysis)
- ST2D: 2D magnetic field analysis (static analysis)
- STAX: Axial symmetry magnetic field analysis (static analysis)
- FQ3D: 3D magnetic field analysis (frequency response analysis)
- FQ2D: 2D magnetic field analysis (frequency response analysis)
- FQAX: Axial symmetry magnetic field analysis (frequency response analysis)
- TR3D: 3D magnetic field analysis (transient response analysis)
- DP2D: 2D magnetic field analysis (transient response analysis)
- DPAX: Axial symmetry magnetic field analysis (transient response analysis)

*2 The FEM coil condition and FEM conductor condition cannot be used independently. They can only be used for an analysis with a circuit.

*3 A slide condition must always be set with a motion condition.

3-4. Material Property for Each Part: Permanent Magnet

3-4-1. Specifying the Magnetization Direction of a Permanent Magnet

There are following 11 patterns of magnetization direction.

If the magnetization direction is already set by the magnetizing tool, you can only confirm the magnetization direction but not change the setting.

- Parallel pattern (circular direction)
- Radial pattern (circular direction)
- Polar pattern (circular direction)
- Axis pattern (circular direction)
- Parallel
- Radial
- Circular
- Spherical
- Parallel pattern (rectilinear)
- Polar pattern (rectilinear)
- Anisotropy by element *Use the magnetization tool

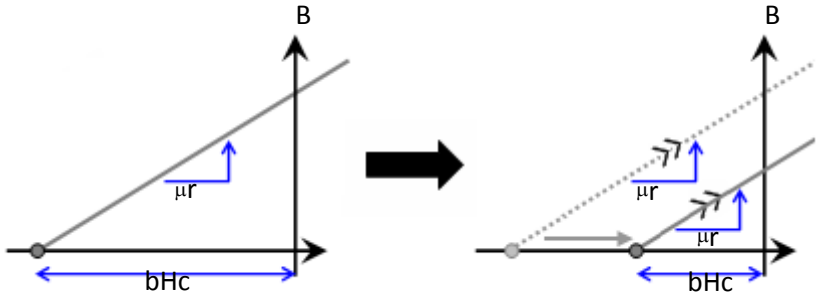
▶▶ Reference

- The direction of magnetization can be selected in the [Magnetization Pattern] dialog box in JMAG-Studio. For more information, see "Material Properties: Magnetizing Direction of Magnets" in Chapter 9, "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information about the magnetization tool in JMAG-Studio, see P.858 "12-2. Magnetization for Magnetic Field Analysis".
- For more information about the magnetization tool in JMAG-Designer, see Chapter 28, "Assigning Materials and Setting Material Properties" in the JMAG-Designer User's Manual.

■ Parallel Pattern (Circular Direction)

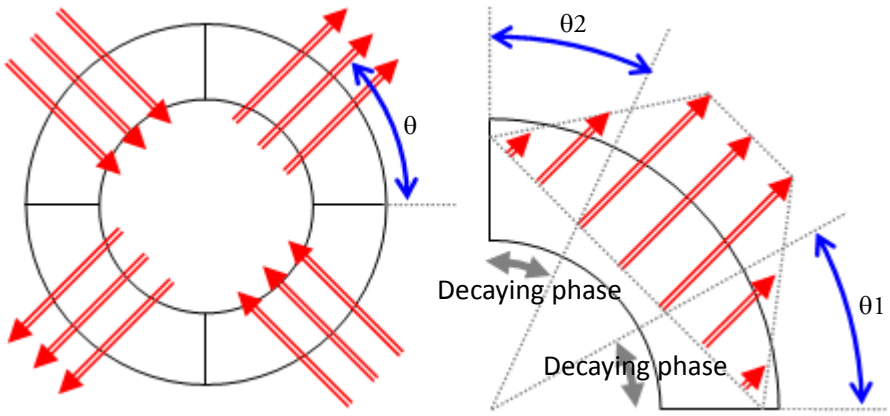
- A circular anisotropic magnetization pattern where the pole having the parallel magnetization vector repeats while reversing itself along the circumference.
- Set the number of magnetic poles per circle.
- The pattern starts from the N pole.
- Specify the location of the first N pole by entering the angle from the X-axis (angle θ shown below) in the [Starting Position θ_0 (deg)] text box.
- The decay in each pole end can be modeled.

  Specify the range of the decay by entering the rising angle and falling angle in the [Angle θ_1 (deg)] and [Angle θ_2 (deg)] text boxes, respectively. A falling angle always lies ahead of a rising angle in the counterclockwise direction.
  With this setting, the coercive force bHc decreases in the decaying regions while the recoil permeability μr is constant.



Before decaying is specified After decaying is specified

- Specify the center of the circular magnetization pattern.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.



A magnet with 4 poles and the start angle specified by θ

- Set the skew.

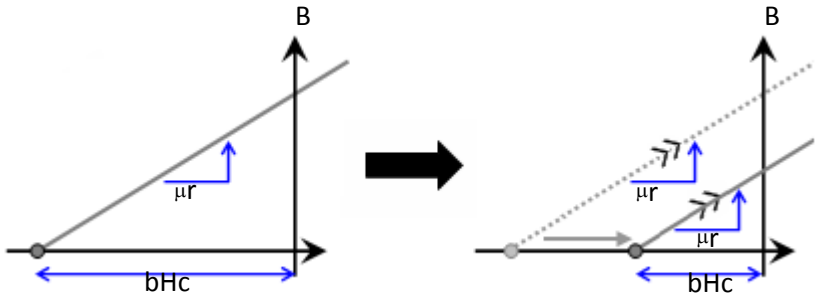
Reference

For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Radial Pattern (Circular Direction)

- A circular magnetization pattern with a radial anisotropy where the pole having the magnetizing vector in the radial direction repeats while reversing itself along the circumference.
- Set the number of magnetic poles per circle.
- The pattern starts from the N pole.
- Specify the location of the first N pole by entering the angle from the X-axis (angle θ shown below) in the [Starting Position θ_0 (deg)] text box.
- The decay in each pole end can be modeled.

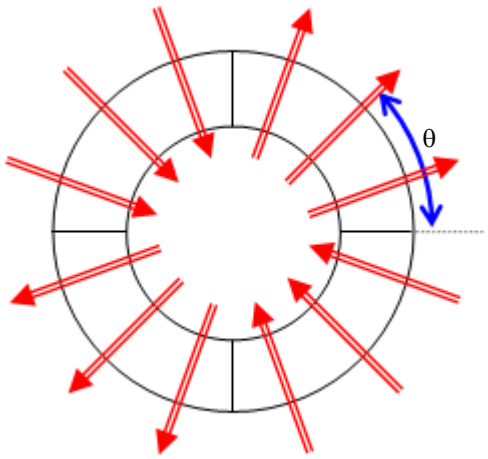
  Specify the range of the decay by entering the rising angle and falling angle in the [Angle θ_1 (deg)] and [Angle θ_2 (deg)] text boxes, respectively. A falling angle always lies ahead of a rising angle in the counterclockwise direction.
  With this setting, the coercive force bHc decreases in the decaying regions while the recoil permeability μr is constant.



Before decaying is specified After decaying is specified

- Specify the center of the circular magnetization pattern.

- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.



A magnet with 4 poles and the start angle specified by θ

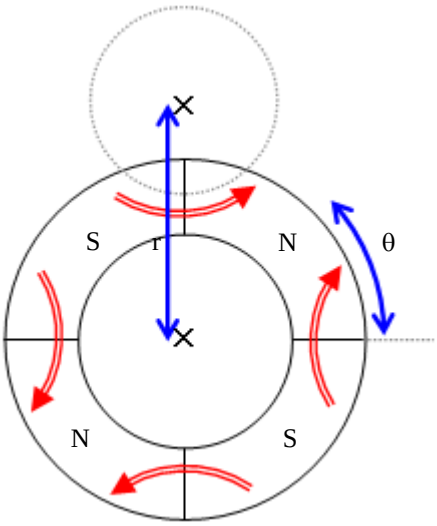
- Set the skew and step skew.

> **Reference**
> For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Polar Pattern (Circular Direction)

- A circular magnetization pattern with a polar anisotropy where the pole repeats while reversing itself along the circumference.
- Set the number of magnetic poles per circle.
- The pattern starts from the N pole.
- Specify the distance between the center of the magnets and the polar anisotropy in the [Radius] box.
- Specify the location of the first N pole by entering the angle from the X-axis (angle θ shown below) in the [Starting Position θ_0 (deg)] text box.
- Specify the center of the circular magnetization pattern.

- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.



A magnet with 4 poles, the start angle specified by θ,
and the radius by r.

- Set the skew and step skew.

Reference

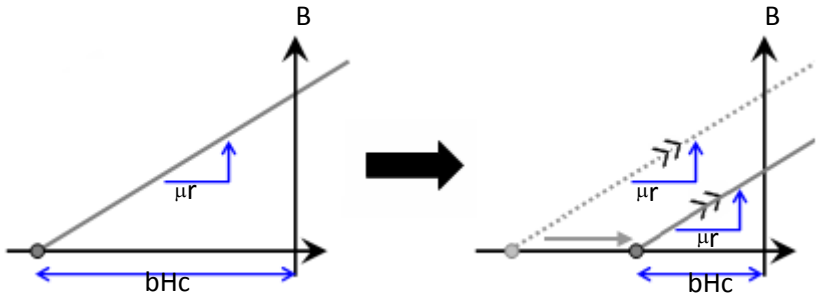
For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Axis Pattern (Circular Direction)

- A circular magnetization pattern with a parallel-to-axis anisotropy where the pole the magnetization vector in the axis direction repeats while reversing itself along the circumference.
- Set the number of magnetic poles per circle.
- The pattern starts from the N pole.
- Specify the location of the first N pole by entering the angle from the X-axis (angle θ shown below) in the [Starting Position θ_0 (deg)] text box.

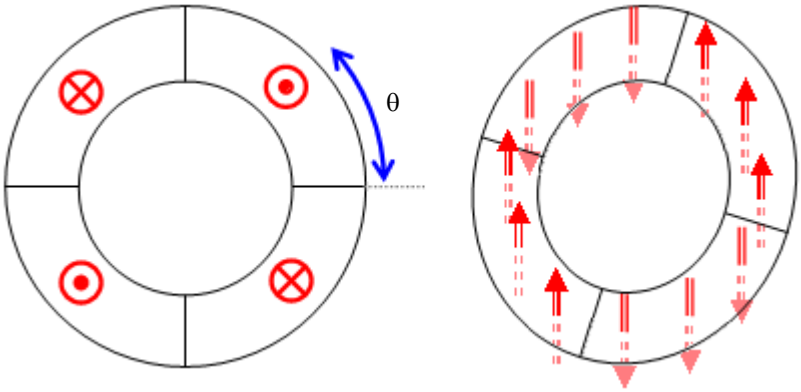
- The decay in each pole end can be modeled.

  Specify the range of the decay by entering the rising angle and falling angle in the [Angle θ $_1$ (deg)] and [Angle θ $_2$ (deg)] text boxes, respectively. A falling angle always lies ahead of a rising angle in the counterclockwise direction.
  With this setting, the coercive force bHc decreases in the decaying regions while the recoil permeability μr is constant.



Before decaying is specified After decaying is specified

- Specify the center of the circular magnetization pattern.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.



A magnet with 4 poles and the start angle specified by θ

- Set the skew and step skew.

▶▶ Reference

For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Parallel

- Magnetization vector that is uniform on the whole material.
- There are two ways to set the direction:
  - Set by vector
  - Set by angle from the X-axis and the Z-axis
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.
- The skew and step skew can be set.

▶▶ Reference
For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Radial

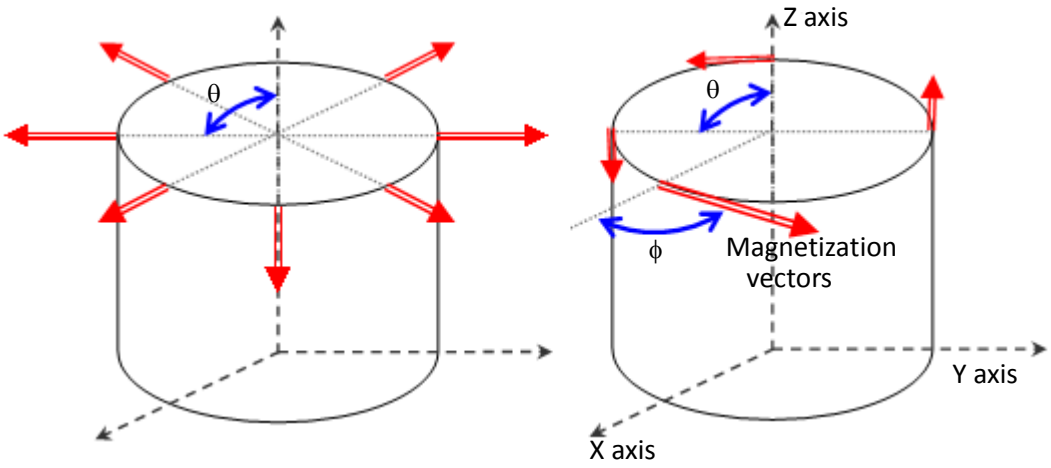
- Radial magnetization vector for the whole material specified as cylindrical.
- Specify the center of the magnetization pattern.
- Radial magnetization vectors can be shifted by specifying the angle from the X- and Z-axes.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.

■ Circular

- Circumferential magnetization vector for the whole material specified as cylindrical.
- Specify the center of the magnetization pattern.
- Radial magnetization vector can be shifted by specifying the angle from the X- and Z-axes.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.

- When setting radial magnetization vectors pointing from the Z-axis of a local coordinate system to the center of each element, the angle from the radial direction, ϕ, is equal to 0 degrees, and the angle from the Z-axis, θ, is equal to 90 degrees.

  When magnetization vectors tangential to circumference are set, the angle from the radial direction, ϕ, is equal to 90 degrees and the angle from the Z-axis, θ, is equal to 0 deg.



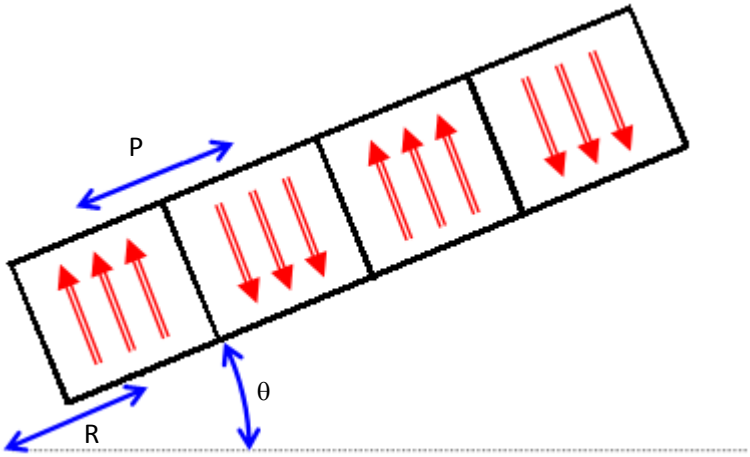
(a) Example of Radial Anisotropy (ϕ= 0, θ= 90)
(b) Example of Circumferential Anisotropy (ϕ = 90, θ = 90)

■ Spherical

- Radial magnetization vectors on the whole material.
- Specify the center of the magnetization pattern.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.

■ Parallel Pattern (Rectilinear)

- A linear magnetization pattern where a pole having the parallel magnetization direction repeats with an anisotropy.
- Enter the distance between the centers of the poles shown as P in the figure below in the [Pitch P (m)] text box.
- The pattern starts from the N pole.
- Specify the magnet orientation by entering the rotation angle from the X-axis, shown below as θ, in the [Elevation from X-axis θ (deg)] text box.
- Enter the distance from the origin to the center of the reference pole, shown below as R, in the [Start R (m)] text box. For JMAG-Designer, define the location of the origin in the [Start Point] text box.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.



P = pitch, R = start position, θ = angle from the X-axis

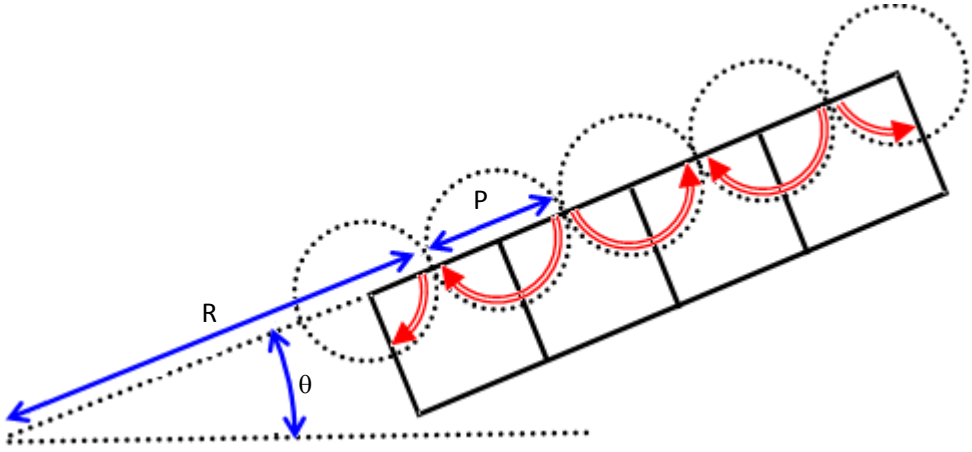
- Set the skew.

▶▶ Reference

For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Polar Pattern (Rectilinear)

- A linear magnetization pattern where a magnet repeats with an anisotropy.
- Enter the distance between the centers of the poles shown as P in the figure below in the [Pitch P (m)] text box.
- The pattern starts from the N pole.
- Specify the magnet orientation by entering the rotation angle from the X-axis, shown below as θ, in the [Elevation from X-axis θ (deg)] text box.
- Enter the distance from the origin to the center of the reference pole, shown below as R, in the [Start R (m)] text box. For JMAG-Designer, define the location of the origin in the [Start Point] text box.
- Define the local coordinate system of the specified center. If not setting this, the global coordinate system is used.



P = pitch, R = start point, θ = angle from the X-axis

- Set the skew.

▶▶ Reference

For information about the skew, see "Setting the Skew for the Magnetization Direction of a Permanent Magnet" on page 88.

■ Anisotropy by Element

- The magnet set by the magnetizing tool has the magnetization vector that changes its direction for each element.
- The setting cannot be created or modified.
- The direction of magnetization vector can only be confirmed.

▶▶ Reference

- For more information about the magnetization tool in JMAG-Studio, see "12-2. Magnetization for Magnetic Field Analysis" on page 858.
- For more information about the magnetization tool in JMAG-Designer, see Chapter 28, "Assigning Materials and Setting Material Properties" in the JMAG-Designer User's Manual.

3-4-2. Setting the Skew for the Magnetization Direction of a Permanent Magnet

The magnetization pattern of a permanent magnet can be skewed (skew magnetization). There are two types of skew magnetization.

- Circular pattern:
  When the parallel pattern (circular direction), radial pattern (circular direction), polar pattern (circular direction), axis pattern (circular direction), or parallel pattern is selected from [Magnetization Pattern].
- Rectilinear pattern:
  When the parallel pattern (rectilinear) or polar pattern (rectilinear) is selected from [Magnetization Pattern].

The permanent magnet can be divided into multiple steps for both patterns, and the magnetization pattern of each step can be skewed (step skew monetization).

■ Equation

- The amount of rotation or translation for the magnetization vector can be calculated using the following equation in the case of the skew magnetization.

$$\theta = \frac{z}{l}\alpha + \beta \qquad (1)$$

  - θ: Amount of rotation or translation for the monetization vector of magnetic material elements
  - z: Z-coordinate of the center of the magnetic material element. The XY-plane of the specified coordinate system is the reference face.
  - l: Stack length of the magnet
  - α: Skew amount
    - Angular difference between the top face and bottom face of the stack length for the circular pattern
    - Distance from the top face to the bottom face of the stack length for the rectilinear pattern.
  - β: Start point on the N pole
    - The X-axis of the specified coordinate system is 0 degrees for the circular pattern.
    - The origin of the specified coordinate system is 0 m for the rectilinear pattern.

- The amount of rotation or translation for the magnetization vector can be calculated using the following equation in the case of the step skew magnetization.

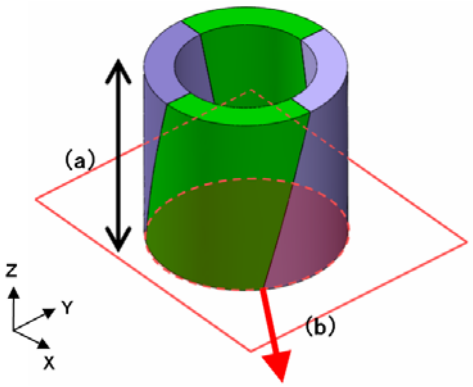
$$\theta = \begin{cases} \left(\frac{1}{2} + n - 1\right)\ \omega + \beta & \text{(Total number of steps: even number)} \\ \left\{\frac{1}{2} + \frac{1}{2}(2n-1)\right\}\omega + \beta & \text{(Total number of steps: odd number)} \end{cases} \quad (2)$$

  - ω: Amount of rotation or translation for the monetization vector of each step
  - n: Number of steps for the step skew that includes a magnetic material element

$$\omega = \frac{\alpha}{(N-1)} \quad (3)$$

$$n = \mathrm{int}\left(\frac{z}{1}N\right) \quad (4)$$

  - N: Total number of the step skew



■ Circular pattern

- Specify the stack length of the magnet and the skew amount (angle)
- The magnetization pattern is applied to the model with reference to the XY-plane of the specified coordinate system.

  Fig. 1 shows the XY-plane that is positioned at the bottom of the magnet.

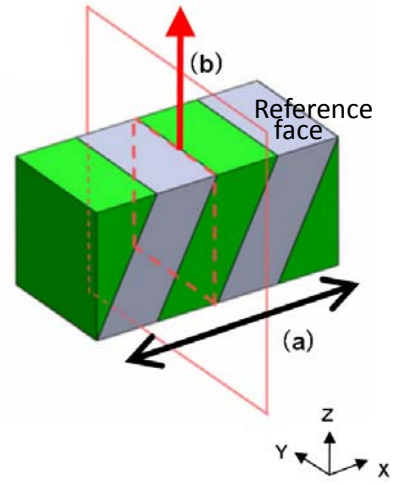


Fig. 1. Skew (XY-plane positioned at the bottom of the magnet)
(a) Stack length (b) XY-plane of the coordinate system

Fig. 2 shows the XY-plane that is positioned at the middle of the magnet.

The position of the face where the magnetization pattern is applied is different from Fig. 1. The start position of the magnetization pattern is different between Fig.1 and Fig 2. However, the stack length and the skew amount (angle) are the same.

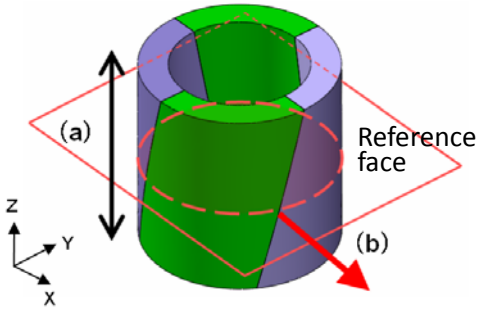


Fig. 2.Skew (XY-plane positioned at the middle of the magnet)
(a) Stack length (b) XY-plane of the coordinate system

- The number of steps needs to be specified to set the step skew. If the number of steps is set to 1, the step skew is not set.

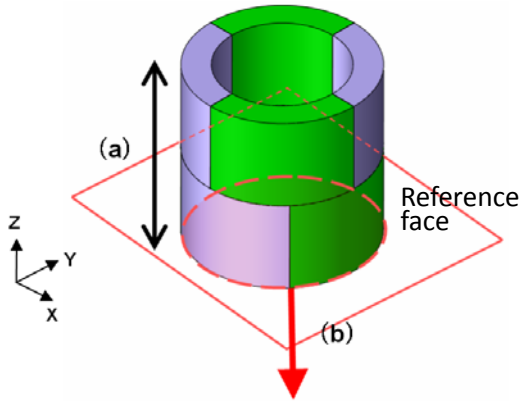
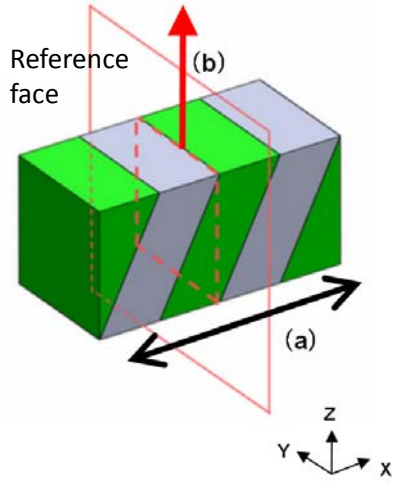


Fig. 3. Step skew (number of steps is 2)
(a) Stack length (b) XY-plane of the coordinate system

■ Rectilinear pattern

- Specify the stack length of the magnet and the skew amount (distance).
- The magnetization pattern is applied to the model with reference to the YZ-plane of the specified coordinate system.



Skew (YZ-plane positioned at the middle of the magnet)
(a) Stack length (b) YZ-plane of the coordinate system

- The number of steps needs to be specified to set the step skew. If the number of steps is set to 1, the step skew is not set.

3-4-3. Using the Magnetizing Tool for a Permanent Magnet (JMAG-Studio)

In JMAG-Studio, the magnetization of a magnet can be mapped by using the magnetizing tool when the [Use Magnetization Program] check box is selected.

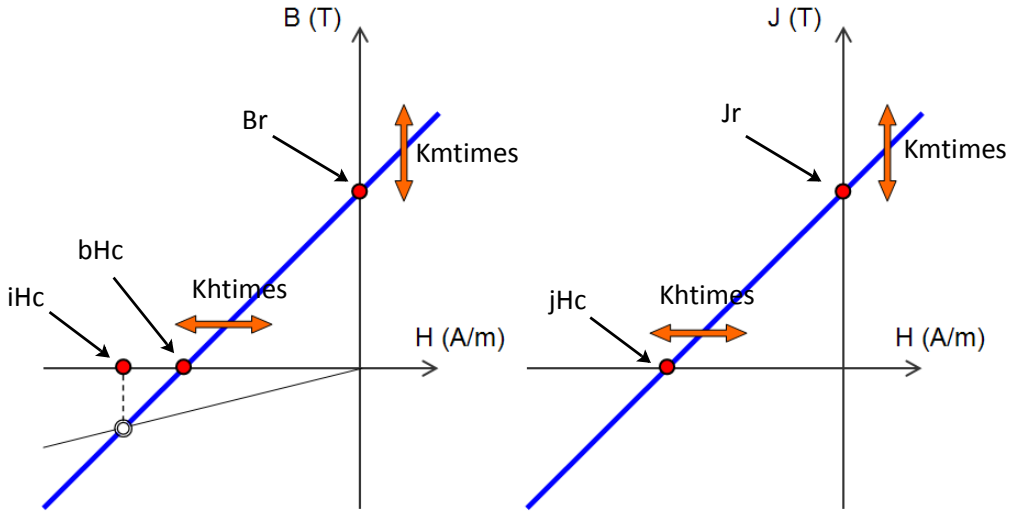
▶▶ Reference

- For more information about the magnetization tool in JMAG-Studio, see P.858 "12-2. Magnetization for Magnetic Field Analysis".
- For more information about the magnetization tool in JMAG-Designer, see Chapter 28, “Assigning Materials and Setting Material Properties” in the JMAG-Designer User's Manual.

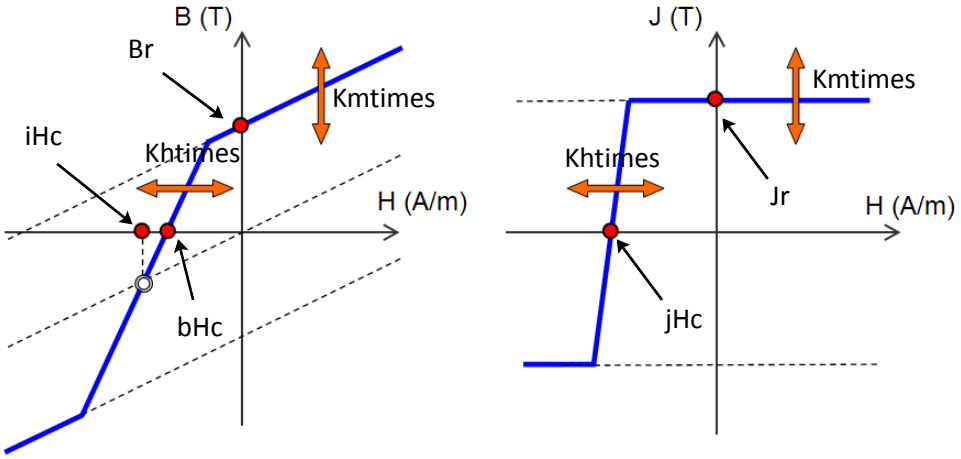
3-4-4. Correcting the Magnetizing Properties of a Permanent Magnet

The residual magnetic flux density or coercive force can be multiplied by the correction value.

■ Linear material



■ **Nonlinear material**

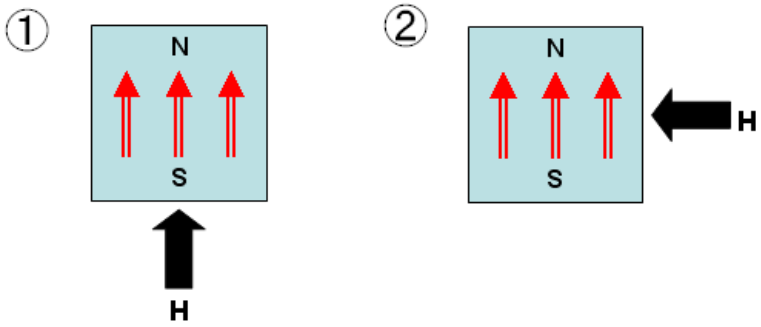


Important **The correction feature and the user subroutine cannot be used together.**

3-4-5. Specifying Isotropy or Anisotropy of a Permanent Magnet

Specify the magnetizing properties when the external magnetic field H is added to the magnet in the direction of normal component.

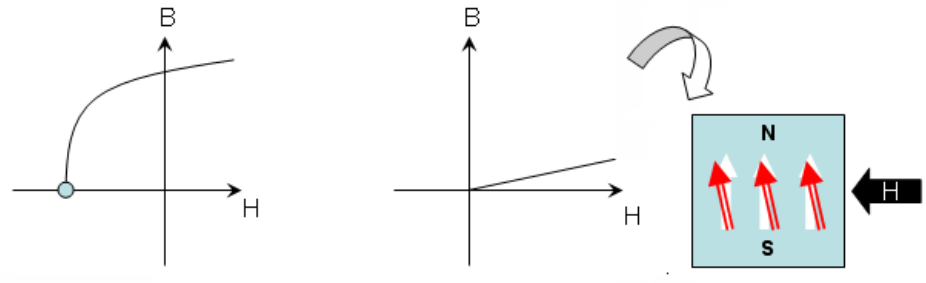
Take two cases 1 and 2, for example, where 1 has an external magnetic field applied in the magnetization direction, and 2 has an external magnetic field applied in the normal direction.



Magnetic field applied in the magnetization direction

Magnetic field applied in the normal direction

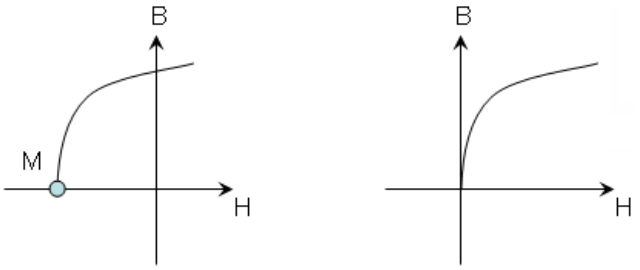
- If selecting [Anisotropic], the nonlinear magnetic property is applied to the components in the magnetization direction. To the components in other directions, the property same as the air is applied.



Magnetic properties
in the magnetization direction

Magnetizing properties
in the normal direction

  - The nonlinear magnetic property is applied to the components parallel to the magnetization direction.
  - The air property is applied to the components normal to the magnetization direction.
- If selecting [Isotropic], the same magnetic property is applied to the components regardless of the magnetization direction.



Magnetizing properties
in the magnetization direction

Magnetizing properties
in the normal direction

  - The same magnetic property is applied to the components in both the magnetization direction and other directions. (The magnetization may occur in the directions other than the magnetization direction.)

MEMO Isotropic cannot be specified for some materials as shown in the following table.

	Isotropic	Anisotropic
Hitachi Metals: NEOMAX (Irreversible) in the material database	X	O
Other magnets in the material database	O	O
Custom irreversible thermal demagnetization magnets *1	X	O
Other custom magnets	O	O

*1: [Custom irreversible thermal demagnetization magnets]
[Nonlinear (Irreversible / Thermal demagnetization)] can be selected in the [Magnetic Properties] tab of the [Material Editor] dialog box in JMAG-Designer.
[Magnetizing Properties (irreversible, any temperature, thermal demagnetization)] can be selected in the [Material Database] dialog box when creating user-defined magnets in JMAG-Studio and a HBP file is used.

MEMO The default setting is [Anisotropic] in Studio 9.1 and Designer 5.0 or later version.

MEMO Isotropic will be specified if you read the data created by a previous version of JMAG into JMAG-Studio 9.1 or JMAG-Designer 5.0.

3-5. Material Property for Each Part: Soft Magnetic Material

3-5-1. Calculating Magnetostrictive Force

Specifies calculation for magnetostrictive force.

- Specify also a lamination direction when magnetostriction properties with anisotropy is set in a part for calculation.

▶▶ Reference

- For more information about magnetostriction properties for materials, see "2-6-3. Specifying the Magnetostriction (Magnetic Field Analysis)" on page 38.
- For more information about specifying lamination direction, see "3-5-2. Specifying Lamination Direction for Magnetic Steel" on page 97.
- For more information about calculating magnetostrictive force, see "3-23. Magnetostriction Calculation (JMAG-Designer)" on page 232.

3-5-2. Specifying Lamination Direction for Magnetic Steel

An analysis using laminated structure of a magnetic steel sheet can be calculated.

- Magnetic flux does not flow well in the lamination direction. To correct the magnetizing properties in the lamination direction, specify the lamination factor. If not correcting the magnetizing properties, set the lamination factor to 100%. If specifying the lamination factor but not specifying the magnetizing properties in the lamination direction, the magnetizing properties are corrected based on the magnetizing properties in the easy-axis direction.
- There are five ways to determine the lamination direction as shown below. From (1) through (4) are available in 2D analysis, 3D analysis and axisymmetric analysis. (5) is available in 2D analysis and axisymmetric analysis.

	Type	Description
(1)	Orthogonal coordinate system	Defines the Z-axis direction of the coordinate system as the lamination direction.
(2)	User subroutine	Specifies the lamination direction using the user subroutine *usrmtx*.
(3)	Cylindrical coordinate system	Defines the r-axis direction of the cylindrical coordinate system as the lamination direction.
(4)	Cylindrical coordinate system	Defines the θ-axis direction of the cylindrical coordinate system as the lamination direction.

	Type	Description
(5)	Angle from X-axis	Specifies the lamination direction by the angle from the X-axis of the global coordinate system.

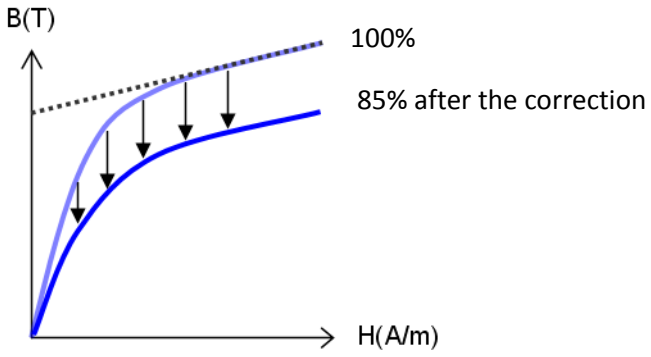
- Specify the lamination direction when anisotropic magnetostriction properties are set for materials specified for calculation in a part. However, this function cannot be used to specify lamination direction by user subroutine.

▶▶ Reference

- For more information about magnetostriction properties for materials, see "2-6-3. Specifying the Magnetostriction (Magnetic Field Analysis)" on page 38
- For more information about using laminated structures of a magnetic steel sheet during eddy current calculation, see "3-6-4. Using Laminated Structures of a Magnetic Steel Sheet during Eddy Current Calculation (JMAG-Designer)" on page 100.

3-5-3. Correcting the Magnetizing Properties of a Soft Magnetic Material

The saturation magnetization can be corrected. The saturation magnetization is multiplied by the correction value for the nonlinear magnetizing properties. The permeability is multiplied by the correction value for the linear magnetizing properties and temperature-dependent permeability.



Important **A magnetic flux density - relative permeability (B–μ) curve cannot be corrected.**

Important **The correction feature and the user subroutine cannot be used together.**

3-6. Material Property for Each Part: Electric Field Analysis

3-6-1. Allowing Eddy Current

If the [Allow Eddy Current] check box is selected, the eddy currents is taken into account in the calculation.

In JMAG-Designer, this check box is in the [Material] settings.

In JMAG-Studio, this check box is in the [Electric] tab in the [Material] dialog box.

3-6-2. Allowing Electric Insulation

In the analysis including the effects of eddy currents, the insulation can be set for the material properties of a conductor. With this setting, the entire surface of the conductor is separated so that the electric insulation can be taken into account.

The following should be noted.

- The insulation cannot be set for the conductors that are on the antiperiodic boundary face. In such case, extend the antiperiodic boundary until it can be a periodic boundary, or expand the partial model to a full model.
- To set the insulation only for a face of a conductor, use an insulation condition.
- This function cannot be used for a FEM conductor in a 3D analysis.
- This function can only be used for a FEM conductor in a 2D analysis. This function has no effect on results when used with other conductors.

▶▶ Reference

- For more information about the FEM conductor condition, see "3-14. FEM Conductor" on page 141.
- For more information about the insulation condition, see “"3-34. Insulation" on page 264”.

3-6-3. Using SIBC (Surface Impedance Method)

In the case of a frequency response analysis dealing with high frequencies, there are cases where the skin depth is extremely thin compared with the thickness of the part, making it difficult to generate a mesh. In such case, select the [Use SIBC] check box.

The following should be noted when using the SIBC.

- The current flows only on the surface of a conductor (surface impedance method), simulating eddy currents. The finer mesh needs not to be generated inside the conductor.
- Shell elements are automatically generated on the outer circumference of the conductor during an analysis. The current density in the analysis results is displayed in the shell elements.

3-6-4. Using Laminated Structures of a Magnetic Steel Sheet during Eddy Current Calculation (JMAG-Designer)

An analysis using laminated structure of a magnetic steel sheet can be calculated, without generating a detailed mesh in the laminated thickness direction.

Select from one of the following specifications.

- Specification of electric properties in lamination direction
  Method of specifying in-plane and lamination direction that differ in electric conductivity/electric resistivity.
- Detailed calculation for electric properties in lamination direction
  Method of calculating laminated steel sheet using internal distribution of laminated steel.
  A loss can be calculated with the same accuracy as the case where mesh is generated for every laminated steel sheet in a 3D analysis. This can be done even if mesh in lamination direction is coarse for a 2D or 3D analysis. This specification is supported for 2D and 3D analyses, but is not supported for axisymmetric models and frequency response analysis.

MEMO A value for lamination surface is displayed as a reference value for an analysis result in distribution of eddy current density when selecting an axial detail calculation.

▶▶ Reference

For more information about specifying lamination direction of magnetic steel, see "3-5-2. Specifying Lamination Direction for Magnetic Steel" on page 97.

3-6-5. Allowing Displacement Current in the Frequency Response Analysis (JMAG-Studio)

In JMAG-Studio, the displacement current can be taken into account in the frequency response analysis (FQ) by selecting non-magnetic material or isotropic magnetic material as the material type, and then specifying the relative permittivity. The electric field (V/m) can be obtained as the results.

The setting of the relative permittivity is not applied unless the [Allow Displacement Current] check box is selected in the [Basic] tab in the [Analysis Control] dialog box. If this check box is not selected, the value entered in the [Relative Permittivity] box will not be used in the analysis.

3-7. Periodic Boundary (Rotation/Translation)

▶▶ Reference

- For more information, see "Periodic Boundary" in Chapter 9 Setting Conditions of Magnetic Field Analysis in the JMAG-Studio User's Manual.
- For more information, see the "Translation Periodic Boundary" and "Rotation Periodic Boundary" sections of Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-7-1. Function

This condition is used to reduce the size of the analysis model when the structure of an analysis object has a rotational or translational periodicity.

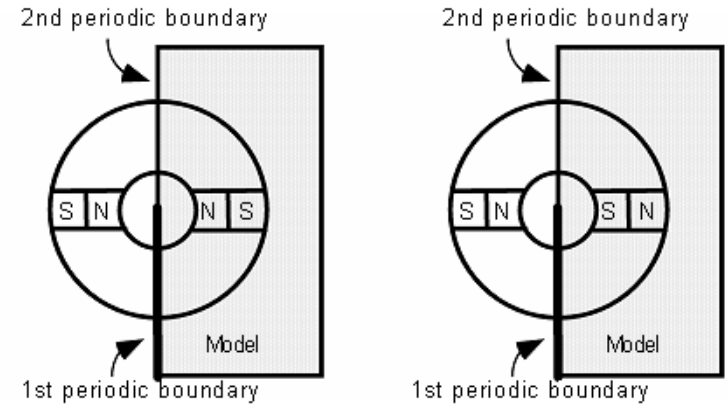
■ Related items

- Electric current
- FEM coil

■ Targets

- Nodes on the first periodic boundary

MEMO The 1st periodic boundary plane refers to the first periodic boundary plane that is in a counterclockwise direction of the two periodic boundary planes in a rotation periodic model.



Periodic boundary Antiperiodic boundary

■ Settings

- Rotational periodicity – translational periodicity flag

  Rotational periodicity, axial reverse rotational periodicity
  - Coordinate of a point passing through the revolution axis
  - Direction of the rotation axis
  - Periodic angle
  - Periodic – antiperiodic flag

    Translational periodicity
  - Direction of translation
  - Periodic distance
  - Periodic – antiperiodic flag

■ Available modules

- All the modules for magnetic field analysis

■ Output items

- N/A

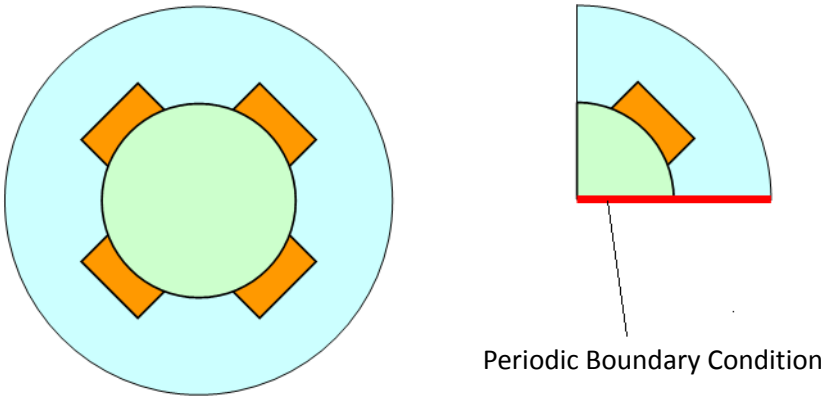
3-7-2. Important Notes Regarding the Use of This Function

- This condition cannot be set more than once for a model.
- For a 3D model, if a boundary condition is set on a face, all of the faces that are on the same plane are automatically regarded as boundary faces.
- For a 2D/axisymmetric model, only the edge where a boundary condition is set is regarded as a boundary edge. Therefore, a boundary condition needs to be set to all the edges on the boundary.
- The boundary condition set on the face or edge is taken into account when the mesh is automatically generated in the gap or the outer boundary of a model using a slide mesh option.

3-7-2-1. Periodic Boundary/Anti-periodic Boundary

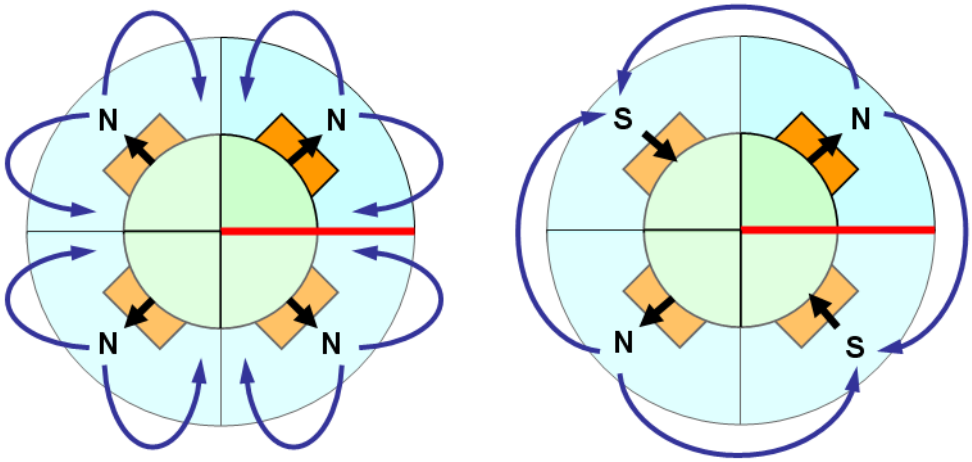
- For the Periodic Boundary condition, select Periodic or Antiperiodic depending on the type of periodicity of the model.

  Take a 4-pole magnet model as an example, the left figure below is the full model and the right figure below is a partial model of the same model. The Periodic Boundary condition is specified on the edge indicated in red in the right hand figure below due to the periodicity of the model.



If selecting Periodic for the Periodic Boundary condition, the magnetic flux flows as shown in the left figure below. The N pole is repeated in the circumferential direction.

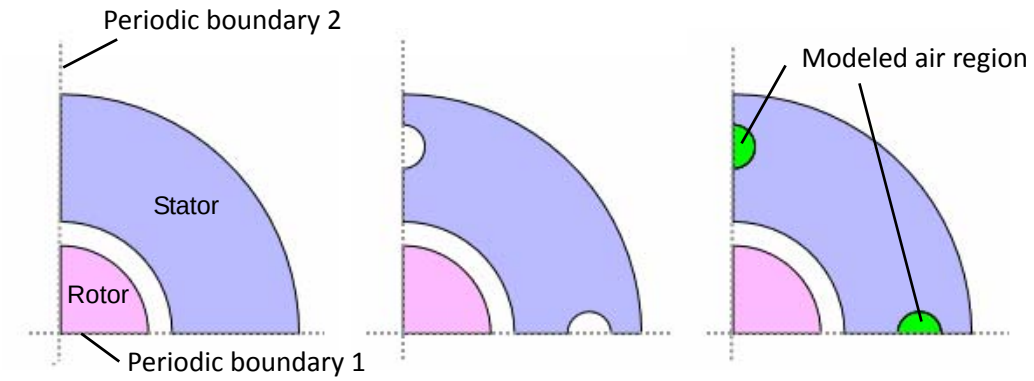
If selecting Antiperiodic for the Periodic Boundary condition, the magnetic flux flows as shown in the right figure below. The N and S poles are repeated alternately in the circumferential direction.



Left: Periodic boundary, Right: Antiperiodic boundary

3-7-2-2. Rotational Periodic Boundary

- "A point on rotation axis" means the coordinates of the center of a rotor. For example, for a model with a periodicity about the Z-axis with its center on the origin, the point should be (0, 0, 0).
- Specify "Direction of rotation axis". For example, if the rotation axis is on the Z-axis, the direction of rotation axis is (0, 0, 1).
- "Periodic angle" is the angle of the rotational periodicity. For example, with a 1/4 model, the periodic angle is 90 degrees.
- In a 2D analysis, the rotation axis of the rotation boundary is the Z-axis.
- If you select [FEM + BEM] as the calculation method in a 2D analysis, the boundary conditions may not be correctly set, causing the calculation to fail depending on the shape and location of the air region. Be sure to create the correct model using the following notes.
  - The analysis can be run if the air region passes through the model from periodic boundary 1 to periodic boundary 2 as shown in Figure A below.
  - The analysis cannot be run if the air regions lie on periodic boundaries 1 and 2 separately as shown in Figure B below.
  - The analysis can be run if the air region is on either periodic boundary 1 or 2 as shown in Figure C below.



- For the periodic boundary, the physical quantity such as flux density remains the same in the amount and direction when rotating the model by the rotation angle.
- For the antiperiodic boundary, the physical quantity such as flux density remains the same while reversing the direction when rotating the model by the rotation angle.

3-7-2-3. Translational Periodic Boundary

- "Translation direction" is the vector representing the direction of translation. For example, if the boundary has a translational periodicity in the X-axis direction, the direction is (1, 0, 0).
- For the translational periodic boundary, enter the periodic distance instead of the periodic angle. For example, if it has a periodicity every 30 cm, the periodic distance is 0.3 m.
- For the periodic boundary, the physical quantity such as flux density remains the same in the amount and direction when moving the model by the translational distance.
- For the antiperiodic boundary, the physical quantity such as flux density remains the same in the amount while reversing its direction when moving the model by the translational distance.

3-7-2-4. Axial Reverse Rotational Periodic Boundary

- "A point on rotation axis" means the coordinates of the center of a rotor. For example, for a model with a periodicity about the Z-axis with its center on the origin, the point should be (0, 0, 0).
- Specify "Direction of rotation axis". For example, if the rotation axis is on the Z-axis, the direction of rotation axis is (0, 0, 1).
- "Periodic angle" is the angle of the rotational periodicity. For example, with a 1/4 model, the periodic angle is 90 degrees.
- The axial reverse rotational periodicity can be used for a claw-pole rotating machine. The rotational periodic boundary condition can also be used. However, if the relation of the flux density is represented as follows, using the axial reverse rotational periodicity can reduce the model size by half.
- The model must be created to have mesh division at the first periodic boundary inverted to that at the second periodic boundary on the rotation axis.

- The relations between flux density and vector potential for a periodic boundary are:

$$B_r(r, \theta, z_0 + z) = -B_r(r, \theta + \theta_0, z_0 - z)$$

$$B_\theta(r, \theta, z_0 + z) = -B_\theta(r, \theta + \theta_0, z_0 - z)$$

$$B_z(r, \theta, z_0 + z) = B_z(r, \theta + \theta_0, z_0 - z)$$

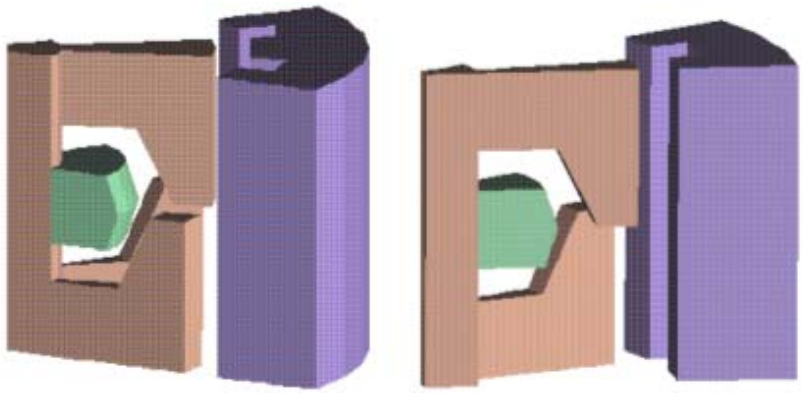
$$A_r(r, \theta, z_0 + z) = A_r(r, \theta + \theta_0, z_0 - z)$$

$$A_\theta(r, \theta, z_0 + z) = A_\theta(r, \theta + \theta_0, z_0 - z)$$

$$A_z(r, \theta, z_0 + z) = -A_z(r, \theta + \theta_0, z_0 - z)$$

$$B = \nabla \times A$$

Where z_0 is the center of the axial coordinates of the nodes with the periodic boundary condition, and θ_0 is the circumferential periodic angle.



Left: Rotational periodic boundary model
Right: Axial reverse rotational periodic model

3-8. Symmetry Boundary/Natural Boundary

▶▶ Reference

- For more information, see "Symmetry Boundary 2D" in Chapter 9, "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Symmetry Boundary" and "Natural Boundary" in Chapter 29"Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-8-1. Function

The symmetry boundary condition is used to specify a symmetry boundary on the analysis model. The natural boundary condition is used to specify a natural boundary on the analysis model.

The magnetic flux flows parallel to the symmetry boundary face and perpendicular to the natural boundary face.

The current needs to be perpendicular to the symmetry boundary.

The current needs not to flow parallel to the natural boundary; however, the natural boundary face needs to be parallel to the current flow direction that is specified by the current condition, FEM coil condition, and FEM conductor condition.

■ Related items

- N/A

■ Settings

- For a symmetry boundary:
  - Symmetry boundary face
  - Vector potential * 2D/axisymmetric analysis of JMAG-Studio
- For a natural boundary:
  - Natural boundary face

■ Available modules

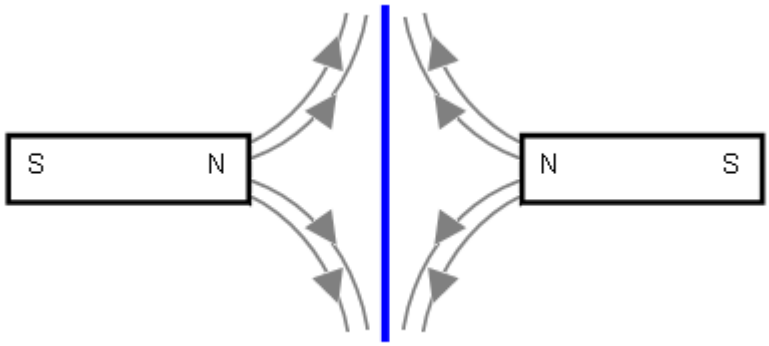
- All the modules for magnetic field analysis

■ Output items

- N/A

3-8-2. Important Notes Regarding the Use of This Function

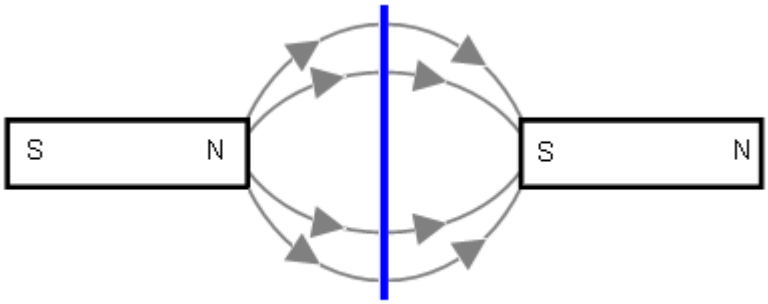
- A symmetry boundary condition is specified on the face on which the same magnetic poles are symmetrically arranged, such as S-to-S poles and N-to-N poles, or on the face by which magnetic field is assumed to be completely shielded by such as magnetic shields.



Boundary face

Symmetry boundary

- Any faces where a symmetry boundary condition is not set are regarded as natural boundary faces to which magnetic flux flows perpendicular.

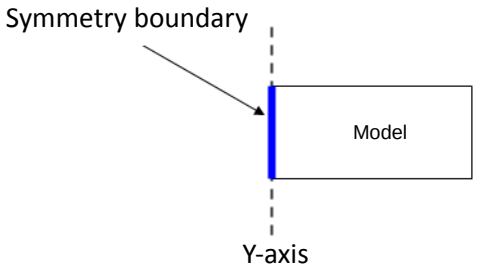


Boundary face

Natural boundary

- For a 2D/axisymmetic model with [FEM] specified as the analysis method, a symmetry boundary condition must be set.
- For a 2D/axisymmetic model, the edge where a boundary condition is set is regarded as a boundary edge. A boundary condition needs to be set for all edges on the boundary.
- For a 2D/axisymmetic model in JMAG-Studio, the vector potential can be specified for a symmetry boundary condition.

- For an axisymmetic model, a symmetry boundary condition needs to be set on the Y-axis regardless of the analysis method (FEM or FEM+BEM).
  If a part is in contact with the Y-axis, a symmetry boundary condition needs to be set on the adjoining area, and the vector potential on the Y-axis needs to be set to zero.



- For a 3D model, a symmetry boundary condition must be set.
- For a 3D model, if a boundary condition is set on a face, all of the faces that are on the same plane are automatically regarded as boundary faces.
- The boundary condition set on the face or edge is taken into account when the mesh is automatically generated in the gap or the outer boundary of a model using a slide mesh option.

3-9. Gap Flux Boundary (JMAG-Designer)

▶▶ Reference

- For more information, see the “Magnetic Field Analysis Using High-Speed Functions in Eddy Current Calculations” in “Chapter 25 Creating a Study by Selecting an Analysis Type” in the JMAG-Designer User's Manual.
- For more information, see the “Gap Flux Boundary Condition” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-9-1. Function

This condition is used to rapidly calculate the eddy currents of magnets based on the results obtained in a 2D magnetic field analysis (transient response analysis).

■ Related items

- N/A

■ Settings

- Magnetic field analysis results file (*.jplot, *.plot)
- Frequency order of the Fourier transform and the range of steps to refer
- The results of 2D analyses is used flag

■ Available modules

- Frequency response analysis (FQ (3D))*Not available in 2D or axisymmetric analyses

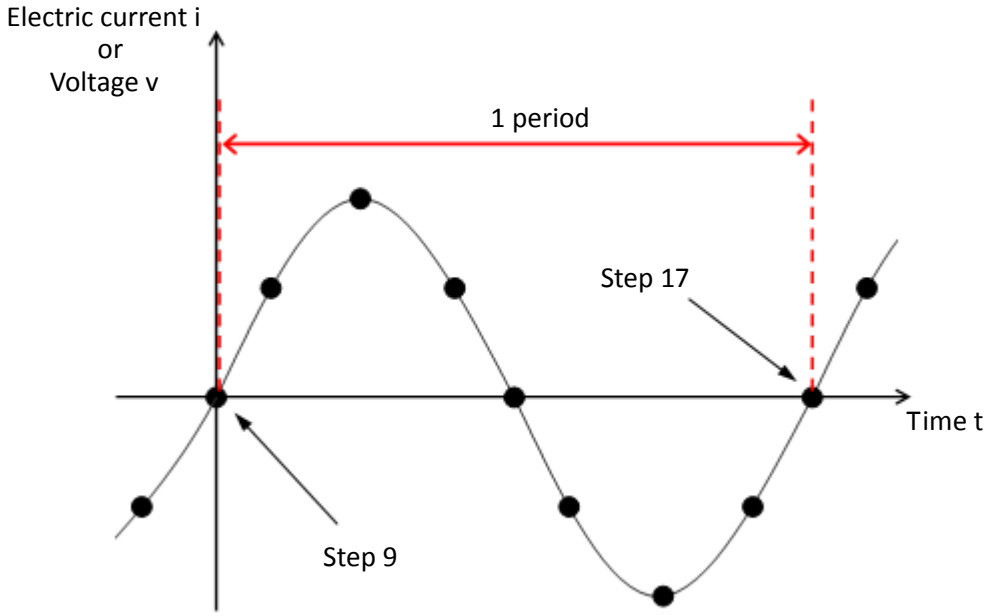
■ Output items

- N/A

3-9-2. Important Notes Regarding the Use of This Function

- This feature is used for solid models. Mesh models are not supported.
- Only model the rotor including the magnets when using this condition.
- Models that have multiple slide faces are not supported.
- Select FEM as the calculation method. FEM+BEM is not supported.

- Select the semi auto mesh as the mesh generation method. Auto mesh is not supported.
- The adaptive mesh feature and mesh features that generate the mesh each analysis step cannot be used.
- The frequency is automatically calculated when this condition is used. The settings specified in the frequency control are not used.
  Frequencies that are automatically calculated are obtained in order by the largest loss value.
- The circuit settings are not used even if they are specified when this condition is used.
- Specify step range that are equivalent to power supply 1 period for fast Fourier transformation (FFT).
  For example, specify '9' in [Start Step] and '17' in [End Step], for a power supply waveform as indicated in the figure below.



3-10. Current

▶▶ Reference

- For more information, see “Current” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Current” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-10-1. Function

This condition is used to specify the current flowing through a coil in terms of current flow region, inflow face, current value, and the number of turns.

■ Related items

- Material properties
- Time/frequency
- Symmetry boundary
- Periodic boundary
- Current density

■ Settings

- Current region
- Current inflow face or inflow direction of the current (for 3D analysis)
- Current value (t) can be set in the following ways.
  - Constant value (ST, TR, FQ, DP)

    MEMO In the frequency response analysis, constant means that the current does not vary with the frequency. Enter the amplitude in the text box. The phase is treated as zero.
  - Point sequence data (ST, TR, FQ, DP)

    The function is defined by a point sequence consisting of time or other properties versus current.
  - Sinusoidal function (TR, DP, FQ)

    The sinusoidal function is defined by amplitude *A*, frequency *F*, and phase *P*.

    $$I(t) = A \cdot \sin(2\pi \cdot F + P \cdot \pi / 180)$$

- Exponential function (TR, DP)

  The exponential function is defined by asymptotic value C_0, amplitude *AMP*, and time constant *T*.

  $$I(t) = C_0 + AMP \cdot \exp(-t/T)$$

- Harmonic (TR, DP)

  Specifies distortion in waveform from a fundamental harmonics and higher harmonics.

- X-axis type for point sequence of current
- X-axis type
- Number of turns
- Eddy current flag
- Periodic point sequence flag for the point sequence data

■ Available modules

All the modules for magnetic field analysis

■ Output items

- Current density (A/m^2)
- Power consumption density (W/m^3)
- Lorentz force density (N/m^3)
- Current value in the current condition (A)
- Flux linkage in the current condition (Wb)

3-10-2. Important Notes Regarding the Use of This Function

- To calculate the Joule loss, specify the electrical conductivity for the materials where the current condition is set.
- If the region where current flows projects outside the model, set the symmetry boundary or periodic boundary condition on the inflow face.
- You need to specify the value of the current flowing in the modeled region. For example, when the current of 1ampere flows in the coil of a full model, if a half of the coil model is created to divide the cross-section area of the inflow face of the current into half, you need to specify 0.5 ampere to the current or specify a half number of turns. Even if a partial model is created from a full model, when the cross-section area of the inflow face of the current in the partial model is the same as that in the full model, you need to set the value of the current and the number of turns to be the same as those of the full model.
- In the frequency response analysis (FQ), the current can be specified by point sequence data, or a sinusoidal function. The point sequence is defined by frequency (Hz) and current value (A). For the current value, enter a complex number considering its phase.

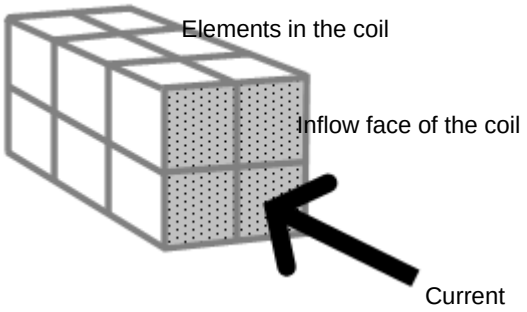
- To set the condition, use the following steps:

  i. Select the coil region through which current flows

  ii. Select the current inflow face

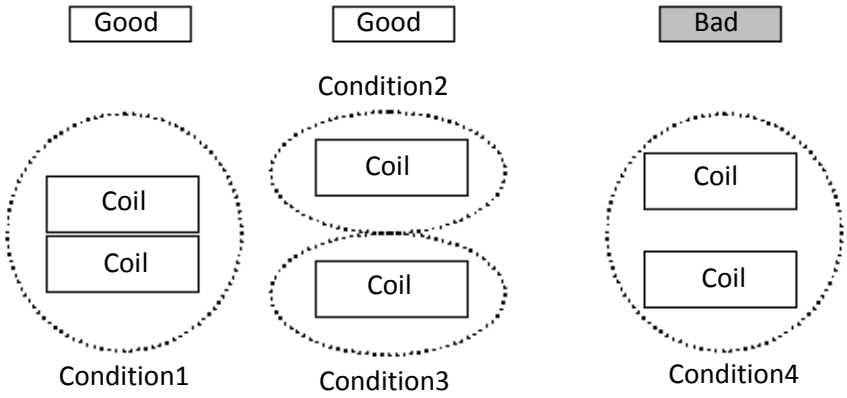
  iii.Set the current value

  For a 2D model, select the same region in steps ii and iii because the region where current flows is the same as the inflow face in a 2D model.



Elements where the current flows and the inflow face

- The current condition needs to be set for each coil region. If the coil region is not continuous, set the condition individually for the separate regions.



Example of the current condition setting

- If setting an inflow face on a ring-shaped coil, choose one side from the two element (or part) faces to set as the inflow face.

Ring shaped coil

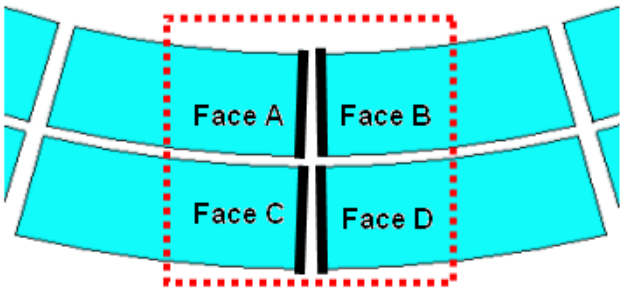


Fig. 1 Enlarged view of ring-shaped coil

As shown in Fig. 1, if selecting the area enclosed by red dotted lines by dragging your mouse, both pairs of elements (or parts) (A and B, C and D) are selected, resulting in the incorrect calculation results.

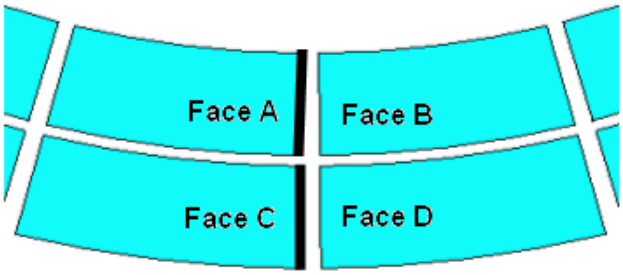


Fig. 2 Enlarged view of ring-shaped coil

As shown in Fig. 2, select only one element of each pair of the elements (or part) (A and C).

- In a 2D analysis or axisymmetric analysis, the current flow direction can be specified to [Up] or [Down]. The [Up] and [Down] are "+Z direction" and "-Z direction" respectively in the XYZ rectangular coordinate system.
  - In a 2D analysis, the direction of the positive current is set in accordance with the XYZ rectangular coordinate system. In other words, the positive current flows in the upward direction (+Z direction) as shown in Fig. 1.
  - In an axisymmetric analysis, the direction of the positive current is set in accordance with the cylindrical coordinate system of RθZ. Since the axis is fixed at the Y-axis, the positive direction of the current is set in accordance with the right screw direction of the Y-axis. In other words, the positive current flows in the downward direction (-Z direction) in the first quadrant of the XYZ rectangular coordinate system as shown in Fig. 2.

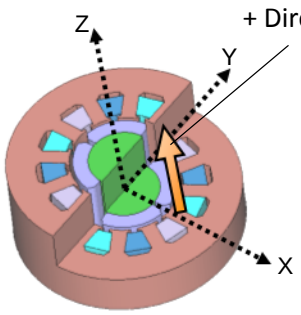


Fig.1

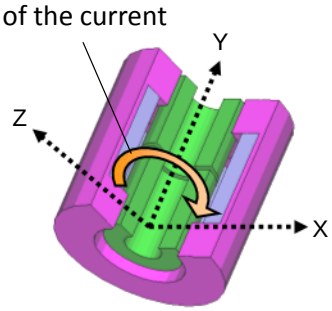


Fig.2

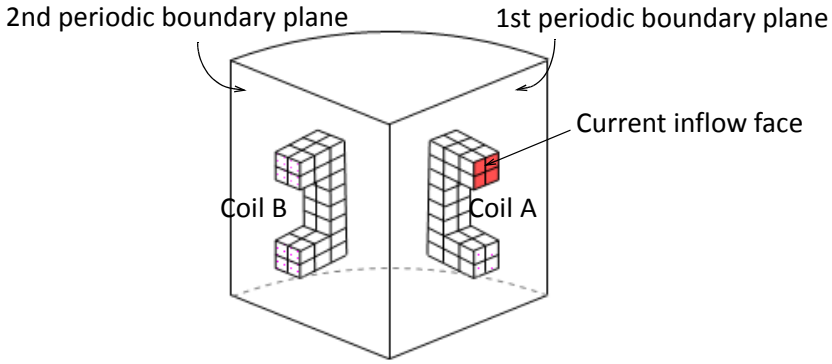
In JMAG-Studio, the current flow direction can be specified under [Direction(2D)] in the [Current] dialog box. In JMAG-Designer, the current flow direction can be specified in the [Current Settings] panel.

- If using the inflow direction vector in 3D analysis, set the start point of the inflow direction vector within the coil region.

3-10-2-1. Setting the Current Condition on a Periodic Boundary

Set one Current condition for the coil connected through the periodic boundaries.

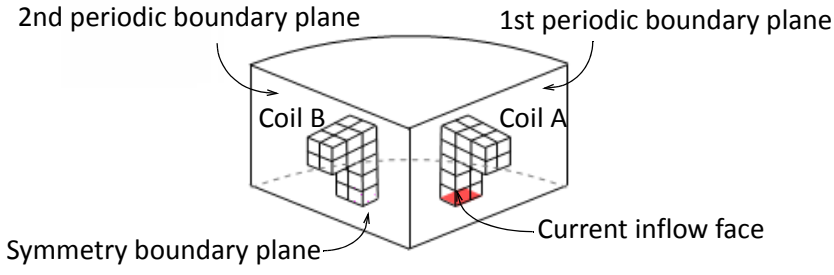
For example, the model shown in Example 1 has Coil A and Coil B that are connected through the periodic boundaries. For this model, select both Coil A and Coil B as the target region of the Current condition. Set the element faces (highlighted in Example 1) on the periodic boundary as the inflow face.



Example 1

MEMO Either Coil A or Coil B can be set as the inflow face.

Example 2 shows a model that is half the model in Example 1 cut lengthwise.

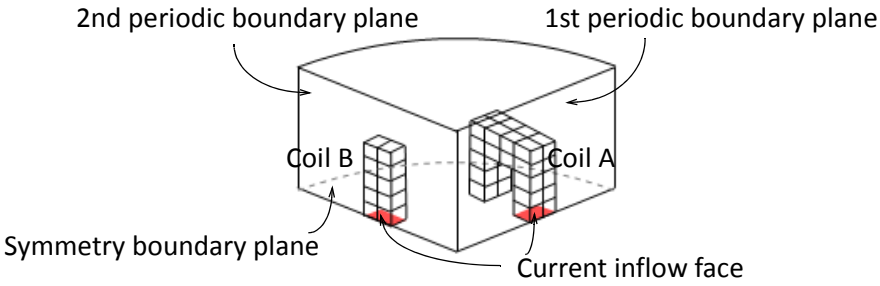


Example 2

Set one Current condition for the coil is connected through the symmetry boundary and periodic boundary as shown in Example 2. Be sure to set the element faces (highlighted in Example 2) on the symmetry boundary as the inflow face.

MEMO Either Coil A or Coil B can be set as the inflow face.

If the coil is divided, set both sections of the coil as the inflow face as shown in Example 3.



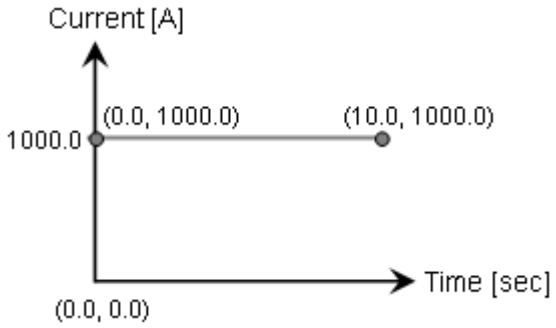
Example 3

3-10-2-2. Point Sequence Data

- For point sequence data, enter two or more points such as (X1, Y1), (X2, Y2). In that case, X1 and X2 must meet the conditions X1<Xmin<X2 and X1<Xmax<X2 for the minimum value, Xmin and the maximum value, Xmax of the X-axis parameters that are referenced in the analysis.
- Time, rotation angle, translation distance, and step IDs can be set for the X-axis of the table, and frequency can also be set in the frequency response analysis (FQ). You can make the setting using a user subroutine.
- When running a static analysis with a current of 1000 A applied to the coil, use the following settings. Note that the type of the X-axis should be time. You can specify the calculation time only from 0 to 10 sec.

Time (sec)	Current (A)
0.0	1000.0
10.0	1000.0

In the static analysis, the normal state time is set to 0.0 seconds, and so the point sequence needs to be created that passes through 0.0 sec.

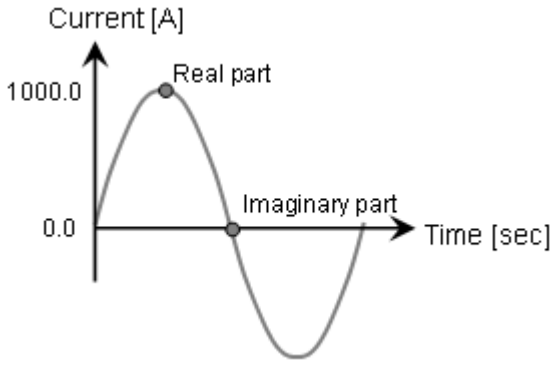


Point sequence data of current in static analysis

- When running a frequency response analysis with current of 1000 A and frequency of 50 Hz applied to the coil, specify the settings as follows.

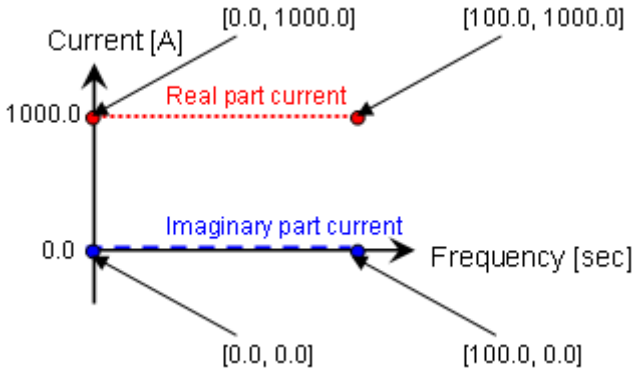
Frequency (Hz)	Current (real part) (A)	Current (imaginary part) (A)
0.0	1000.0	0.0
100.0	1000.0	0.0

In the frequency response analysis, enter the highest current value (real part) and a current value at the phase shifted by 90 degrees from the highest point (imaginary part).



Input current in frequency response analysis

For the frequency response analysis, a point sequence needs to be created to include 50 Hz as the frequency of 50 Hz is used.



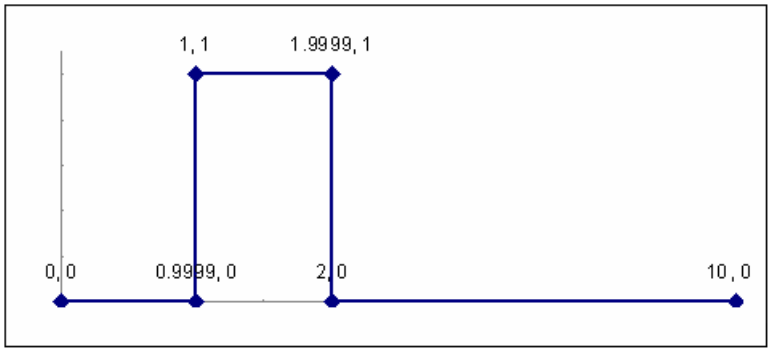
Point sequence data of current in frequency response analysis

- The pulse can be represented by a table as shown below.

Example

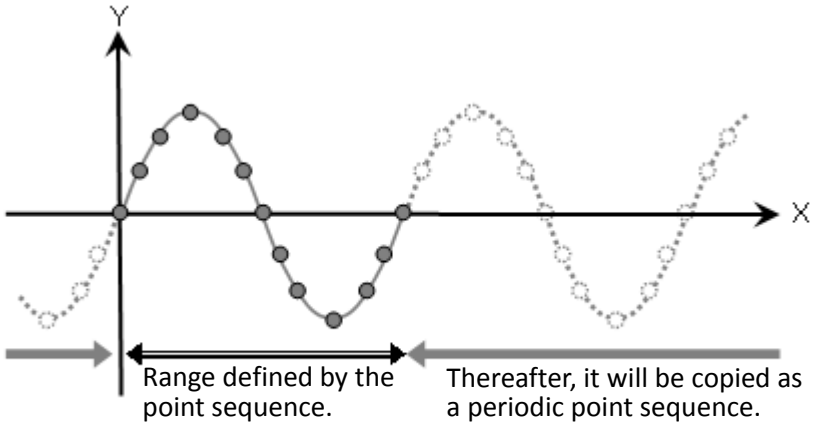
	Time (sec)	Current (A)
1	0.0	0
2	0.9999	0
3	1.0	1
4	1.9999	1
5	2.0	0
6	10	0

A pulse at a time point cannot be defined twice. The rise time and fall time are expressed by slightly shifting the time. The pulse data in the above table can be represented in a diagram as follows.



MEMO The calculation time is determined by the Step condition for JMAG-Studio, and by the study properties for JMAG-Designer. Please note that the calculation time is not defined by the table of the Current condition.

- When the [Periodic Point Sequence] check box is selected, in the case of an X-axis value exceeding the range defined by the point sequence during the calculation, the point sequence will be copied periodically and used in the analysis.



If the [Periodic Point Sequence] check box is selected:

When the [Periodic Point Sequence] check box is cleared, a warning will be output to the solver report if an X-axis value that exceeds the range defined by the point sequence is referenced in the calculation.

3-10-2-3. Flux Linkage

- Magnetic flux (Wb) and current value (A) can be output as step values.
- The flux linkage is the product of the integration of vector potential A in the current direction and the number of turns T. In 2D analysis, the current flows in the Z-axis direction.

$$\Phi = \int ATdll$$

MEMO The flux linkage for the Current condition may be multiplied by a conversion factor according to the full model conversion function. It will not be affected by the circuit conversion function. For more information, see P.283 "3-40. Full Model Conversion".

3-10-2-4. Eddy Current (JMAG-Studio)

- In the [Eddy Current] group box, you can specify whether to generate eddy currents in the area where the current condition is set.
  If selecting the [Allow] radio button, set the electrical conductivity for the material property of the area with the current condition set and select the Allow Eddy Current check box. All these settings are necessary to generate eddy current.

  MEMO The electrical conductivity cannot be specified for a coil. Specify a nonmagnetic material for a coil first, and then set the electrical conductivity.
- With this option, the total amount of current is smaller than the specified amount of current because eddy currents are generated in the direction opposite to the current flow specified for the Current condition.
- Use the FEM Conductor condition to specify the amount of current flowing through a coil, including the eddy currents.

  ▶▶ Reference

  For more information about the FEM Conductor, see P.141 "3-14. FEM Conductor".

3-11. Current Density

▶▶ Reference

- For more information, see “Current Density” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Current Density” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-11-1. Function

This condition specifies the electric current density when, for example, a current flows into a coil. An external current can be set for a beam element.

■ Related items

- Symmetry boundary/Natural boundary
- Current

■ Settings

- Current density
- Beam element in which external current flows
- Current density J (t) can be set in the following ways.
  - Constant value (ST, TR, FQ, DP)

    MEMO In the frequency response analysis, constant means that the current density does not vary with the frequency. Enter the amplitude in the text box. The phase is treated as zero.
  - Point sequence data (ST, TR, FQ, DP)

    A function is defined by a point sequence that consists of time or other parameters and current density.
  - Sinusoidal function (TR, DP, FQ)

    A sinusoidal function is defined by amplitude *A*, frequency *F*, and phase *P*.

    $I(t) = A \cdot \sin(2\pi \cdot F + P \cdot \pi / 180)$
  - Exponential function (TR, DP)

    An exponential function is defined by asymptotic value C_0, amplitude *AMP*, and time constant *T*.

    $I(t) = C_0 + AMP \cdot \exp(-t/T)$

- Harmonic (TR, DP)

  Specifies distortion in waveform from a fundamental harmonics and higher harmonics.

- Direction vector along which current density flows
- X-axis type for point sequence of current density
- Eddy current flag
- Periodic point sequence flag of point sequence

■ Available modules

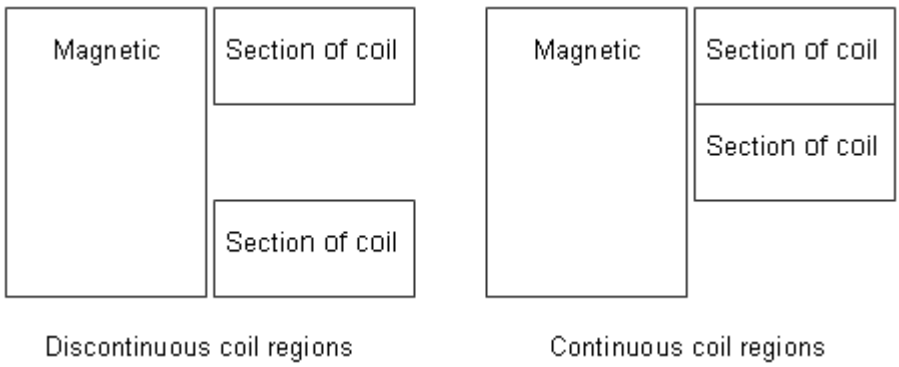
- All the modules for magnetic field analysis

■ Output items

- Current density (A/m^2)
- Joule loss density (W/m^3)
- Lorenz force density (N/m^3)

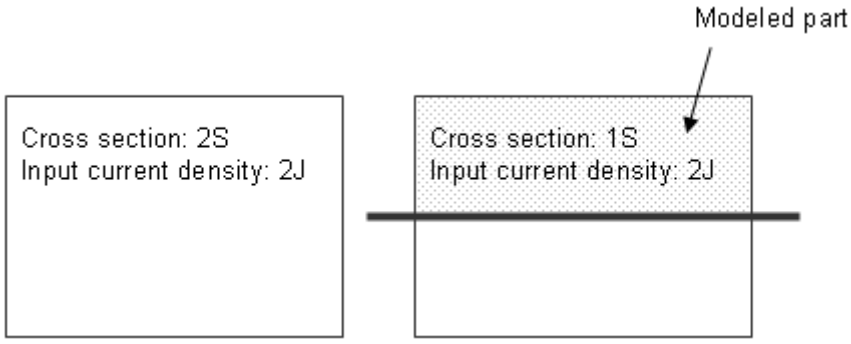
3-11-2. Important Notes Regarding the Use of This Function

- If the current is specified by the current density condition, the results may be incorrect depending on the model due to the divergence of the solutions. To avoid this, use the current condition to specify the current.
- A current density condition can be set on discontinuous coil regions. For both of the following cases, a single current density condition can be set on the coil regions.



Discontinuous coil regions

- Even if the cross section of the coil in the model is a half of the actual coil cross section, the current density to be set should be the same as that of the full model. The current density can be obtained from the coil cross section area and the input current.



Setting of current density.

- In a 2D analysis or axisymmetric analysis, the current flow direction can be specified to [Up] or [Down]. The [Up] and [Down] are "+Z direction" and "-Z direction" respectively in the XYZ rectangular coordinate system.
  - In a 2D analysis, the direction of the positive current is set in accordance with the XYZ rectangular coordinate system. In other words, the positive current flows in the upward direction (+Z direction) as shown in Fig. 1.
  - In an axisymmetric analysis, the direction of the positive current is set in accordance with the cylindrical coordinate system of RθZ. Since the axis is fixed at the Y-axis, the positive direction of the current is set in accordance with the right screw direction of the Y-axis. In other words, the positive current flows in the downward direction (-Z direction) in the first quadrant of the XYZ rectangular coordinate system as shown in Fig. 2.

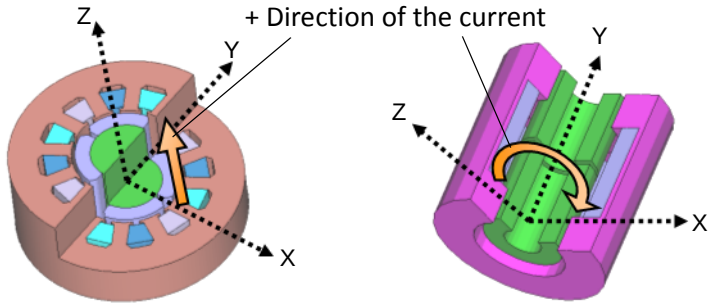
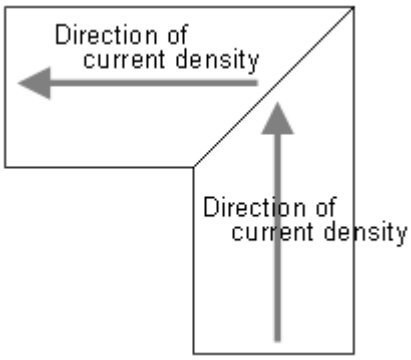


Fig.1 Fig.2

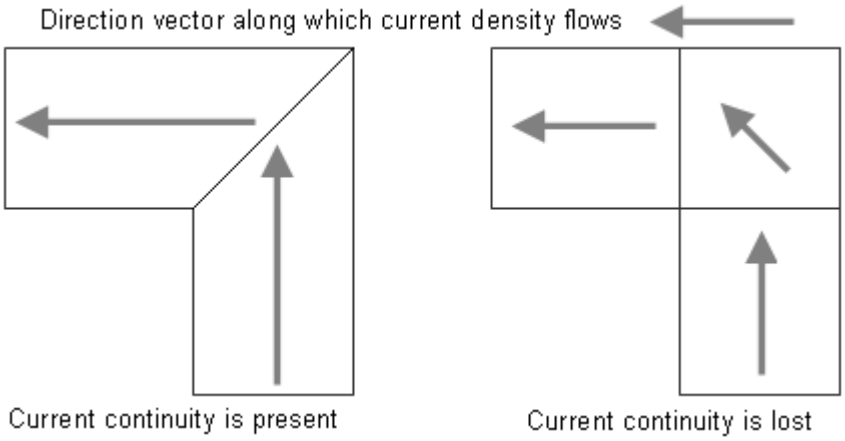
In JMAG-Studio, the current flow direction can be specified under [Current Density Vector] in the [Current Density] dialog box. In JMAG-Designer, the current flow direction can be specified in the [Current Density Settings] panel.

- The direction vector along which current flows can be specified. If there are two or more directions such as a magnetic material with coil wound around it, the vector should be set individually.



Direction vector of current density

- In a 3D analysis, solutions may not be obtained if the current is not continuous.



Continuity of current

3-11-2-1. Eddy Current (JMAG-Studio)

- Eddy currents are generated when a value other than zero is set for the current conductivity, and the [Allow] radio button is selected in the [Eddy Current] group box.

  In that case, since eddy currents are generated to cancel out the values of specified current density, the current that flows in the coil is smaller than the specified value.

3-11-2-2. External Current (JMAG-Studio)

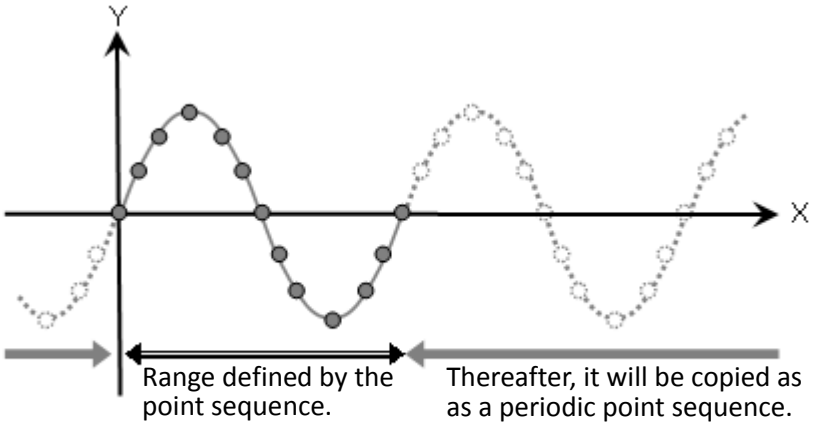
- The external current can be used for 3D analysis only.
- To set an external current, create a coil with beam elements and set a current for the beam elements. The value set is recognized as the current value. Select [External Current (Only 3D)] in the [Eddy Current] group box.
- The external current flows in the incidence direction of the beam elements.
- Parts where an external current flows cannot be represented by a partial model. For example, even if creating a 1/4 model, beam elements must be of the full model.

Table 1 Eddy current flag and coil's electrical conductivity

Eddy current flag	Electrical conductivity of coil	Lorentz force	Joule heat
Not generated	= 0	Not calculated	Not calculated
Not generated	≠ 0	Calculated	Calculated
Generated	= 0	Not calculated	Not calculated
Generated	≠ 0	Calculated	Calculated
External current	= 0	Not calculated	Not calculated
External current	≠ 0	Calculated	Calculated

3-11-2-3. Point Sequence Data

- For point sequence data, enter two or more points such as (X1, Y1), (X2, Y2). In that case, X1 and X2 must meet the conditions X1<Xmin<X2 and X1<Xmax<X2 for the minimum value, Xmin and the maximum value, Xmax of the X-axis parameters that are referenced in the analysis.
- Time, rotation angle, translation distance, step IDs can be set for the X axis of the table, and frequency can also be set in the frequency response analysis (FQ). You can make the setting using a user subroutine.
- When the [Periodic Point Sequence] check box is selected, in the case of an X-axis value exceeding the range defined by the point sequence during the calculation, the point sequence will be copied periodically and used in the analysis.



If the [Periodic Point Sequence] check box is selected:

When the [Periodic Point Sequence] check box is cleared, a warning will be output to the solver report if an X-axis value that exceeds the range defined by the point sequence is referenced in the calculation.

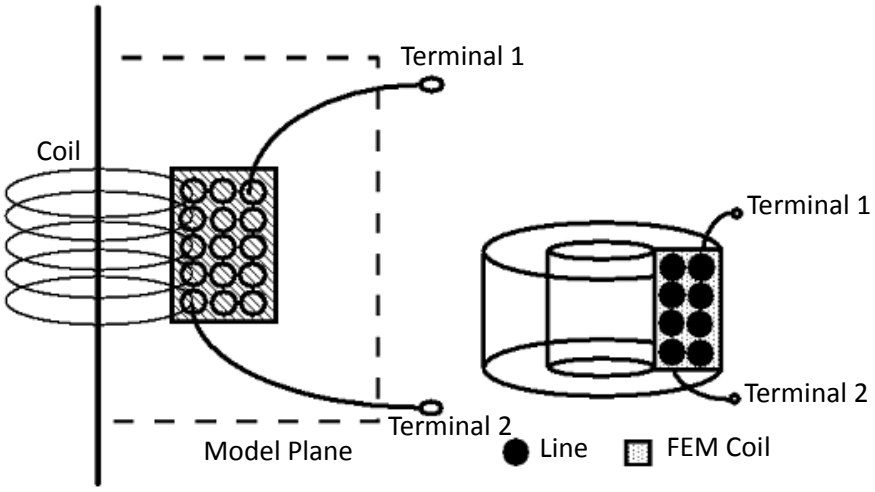
3-12. FEM Coil

Reference

- For more information, see “FEM Coil” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “FEM Coil” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-12-1. Function

When a model includes a coil, currents flowing through the coil can be calculated by giving the voltage of both terminals. Inversely the voltage generated by the flow of currents can be obtained.



FEM coil (axially-symmetric analysis) FEM coil (2D analysis)

■ Related items

- Group (FEM Coil) (140 page)
- External Circuit (147 page) *Mandatory
- Full Model Conversion (283 page)
- Circuit Conversion (286 page)

■ Settings

- Group ID of FEM coil
- Up and down direction flags *In the case of 2D and axially-symmetric analysis

■ Available modules

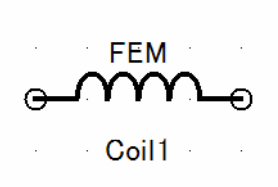
- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

■ Output items

- Current density (A/m^2)
- Current in circuit components (A)
- Electric potential at circuit terminals (V)
- Joule loss density (W/m^3)
- Current value of FEM coil (A)
- Flux linkage of FEM coil (Wb)

3-12-2. Important Notes Regarding the Use of This Function

- In an analysis, it is assumed that a nearly uniform current flows in a specified coil. When an analysis is conducted that includes the generation of an eddy current distribution (such analysis to study the inside of a conductor) inside a specified space, use an FEM conductor.
- Multiple FEM coil conditions can be set for one model.
- To use this function, conditions can be set when an external circuit is used. As the external circuit, one comprised of resistances, coils, capacitors, and switches can be connected. One FEM coil condition corresponds to one FEM coil component (151 page) in the circuit component list.

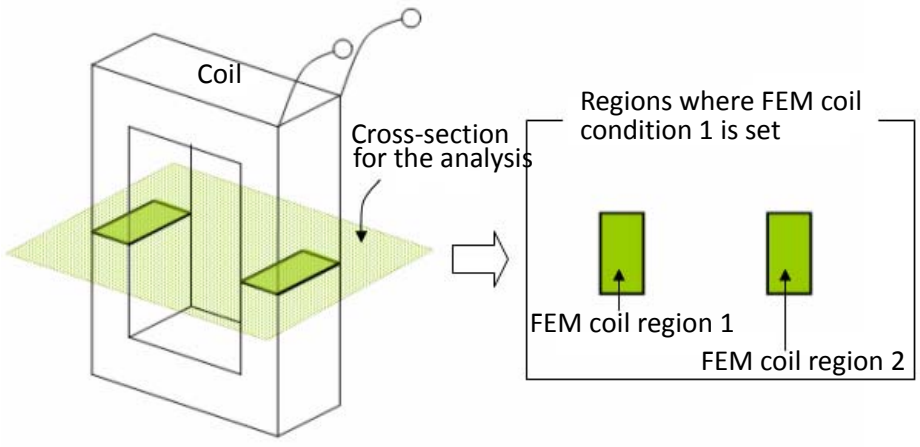


FEM coil component in the circuit

MEMO The number of turns and the resistance values can be set for the FEM coil component in the circuit component list.

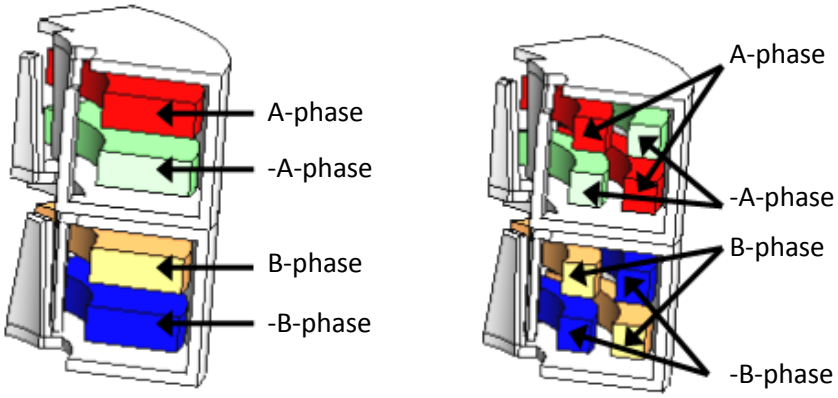
- Procedures for setting and notes comply with the current conditions.
- The electrical conductivity of elements included in the FEM coil is set to 0.0 (1/(ohm・m)).

- One FEM coil condition can be set for multiple coil regions.



Example of how to group coils (for 2D analysis)

- The FEM coil conditions to be included in an FEM coil group must be set on the continuous regions. If FEM coil regions are discontinuous, put the FEM coil conditions in separate FEM coil groups.



In the figure above, the left model has one coil region per phase and the right model has the divided coil regions per phase. For example, the coils in the A-phase in the right model is assigned to two different FEM coil groups separately. In this case, the groups should be connected in series. Also, the number of turns contained in one inflow-plane needs to be set for the FEM coil component. The right model has half the number of turns of the left model.

- In a 2D analysis or axisymmetric analysis, the current flow direction can be specified to [Up] or [Down]. The [Up] and [Down] are "+Z direction" and "-Z direction" respectively in the XYZ rectangular coordinate system.
  - In a 2D analysis, the direction of the positive current is set in accordance with the XYZ rectangular coordinate system. In other words, the positive current flows in the upward direction (+Z direction) as shown in Fig. 1.
  - In an axisymmetric analysis, the direction of the positive current is set in accordance with the cylindrical coordinate system of RθZ. Since the axis is fixed at the Y-axis, the positive direction of the current is set in accordance with the right screw direction of the Y-axis. In other words, the positive current flows in the downward direction (-Z direction) in the first quadrant of the XYZ rectangular coordinate system as shown in Fig. 2.

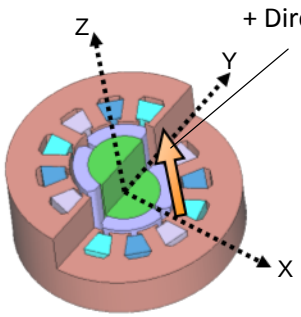


Fig.1

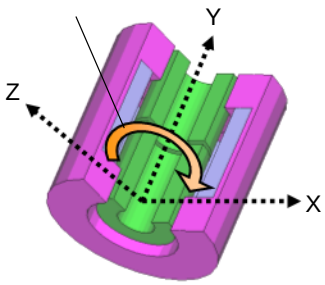
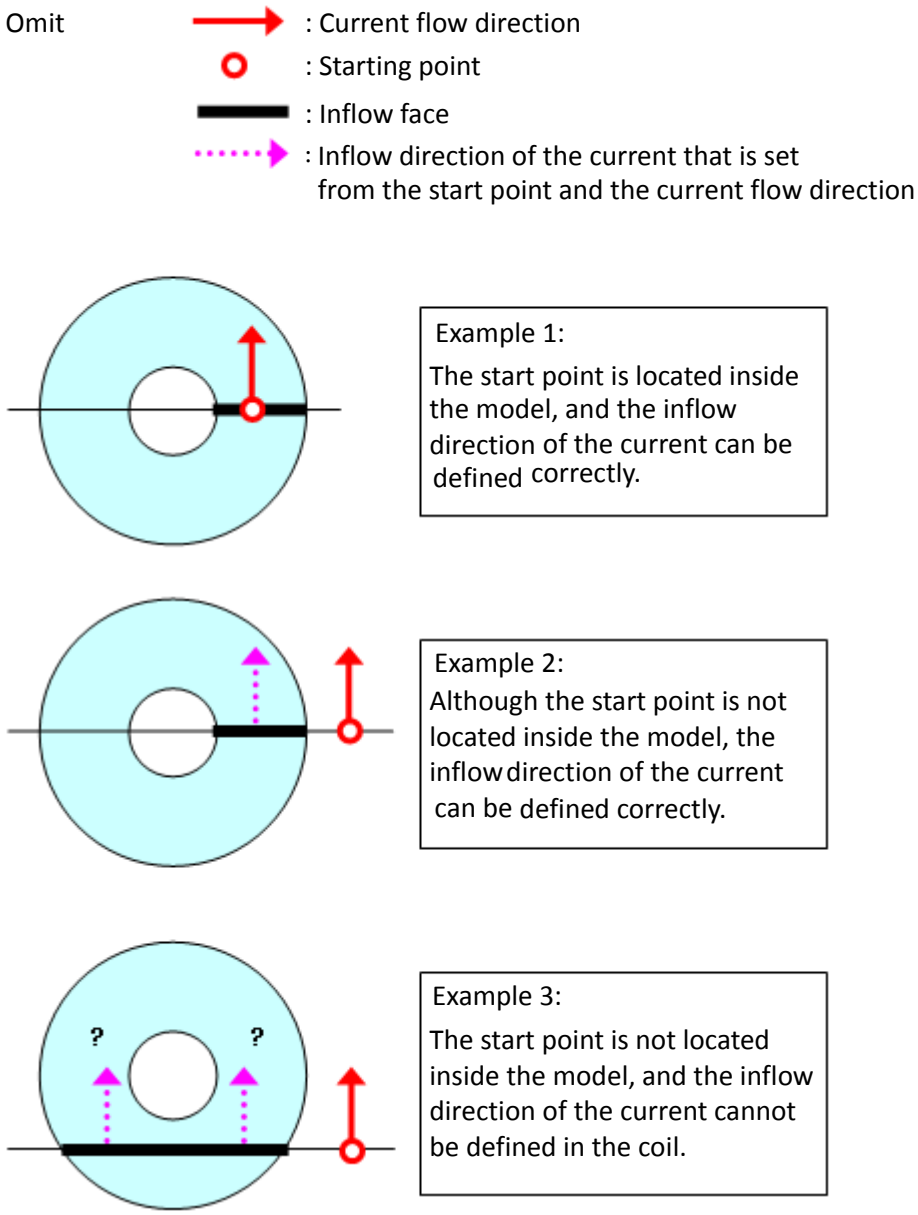


Fig.2

In JMAG-Studio, the current flow direction can be specified under [Direction (2D)] in the [FEM Coil Region] dialog box. In JMAG-Designer, the current flow direction can be specified in the [FEM Coil Settings] panel.

- In a 3D analysis, if the current flow direction is specified with the vector, a face that passes through the start point and that is perpendicular to the current flow direction is defined as an inflow face of the current. Therefore, the start point of the vector should be specified in a position inside the coil, if possible. If the start point is not specified inside the coil, the inflow direction set from the inflow face of the current cannot be defined, and there is possibility of causing errors in the calculation.


Omit
: Current flow direction
: Starting point
: Inflow face
: Inflow direction of the current that is set from the start point and the current flow direction
Example 1:
The start point is located inside the model, and the inflow direction of the current can be defined correctly.
Example 2:
Although the start point is not located inside the model, the inflow direction of the current can be defined correctly.
?
?
Example 3:
The start point is not located inside the model, and the inflow direction of the current cannot be defined in the coil.

Start point and current flow direction

- In a 2D analysis, the length in the Z-direction can be specified. When the length in the Z-direction is not specified, the thickness of a model is treated as the length in the Z-direction.
  The length in the Z-direction is used to determine the induced voltage in the coil. For example, suppose that a magnet is placed at the counter position of a steel sheet wrapped around a coil. If the thickness of the magnet is larger than the thickness of the sheet, magnetic flux generated by the magnet is interlinked with that of the coil. When the length of the coil is longer than that of the sheet, the additional contribution of magnetic flux due to the coil extending beyond the length of the sheet must also be taken into account. In that case, since the thickness of the model (thickness of steel sheet) must not be treated as the length of the coil, the length in the Z-direction needs to be specified. Note that, in 3D analysis and axisymmetric analysis, the length in the Z-direction is ignored.

3-12-2-1. Flux Linkage

- Magnetic flux (Wb) and current value (A) can be output as step values.
- The flux linkage is determined by multiplying the current flow direction line integration of the vector potential by the number of turns *T*.

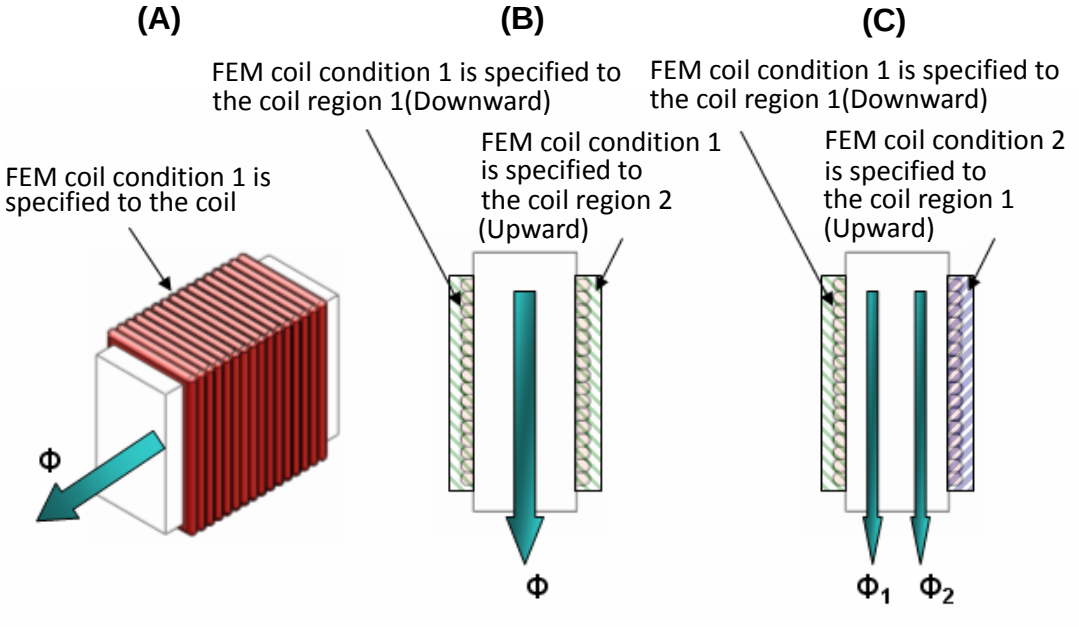
$$\Phi = \int \mathrm{ATdll}$$

- Note that the calculation of flux linkage varies depending on the settings of the FEM coil conditions.

  Fig. A shows a 3D model with a magnetic material wound with coil. The arrow indicates flux linkage. Fig. B and Fig. C show the cross-sections of 2D models created from the 3D model in Fig. A.

  As shown in Fig.B, if one FEM coil condition is specified to two FEM coil regions, the magnetic flux for the FEM coil condition is treated as the flux linkage.

  As shown in Fig. C, if one FEM coil condition is specified to each FEM coil region, the total amount of the magnetic flux for each FEM coil condition is treated as the flux linkage.


(A)
(B)
(C)
FEM coil condition 1 is specified to the coil region 1(Downward)
FEM coil condition 1 is specified to the coil region 1(Downward)
FEM coil condition 1 is specified to the coil
FEM coil condition 1 is specified to the coil region 2 (Upward)
FEM coil condition 2 is specified to the coil region 1 (Upward)
Φ
Φ
Φ1
Φ2

(A) 3D model
(B) 2D model (one FEM condition for two regions)
(C) 2D model (one FEM condition for each region)

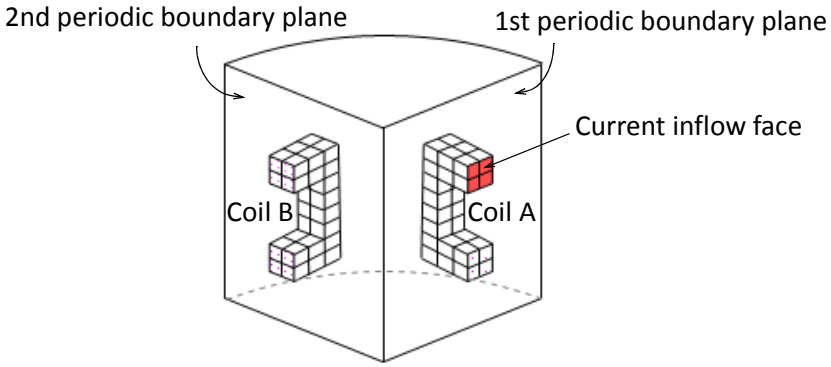
MEMO The flux linkage for the FEM Coil condition may be multiplied by a conversion factor according to the full model conversion function. It will not be affected by the circuit conversion function.

▶▶ Reference
For more information, see P.283 "3-40. Full Model Conversion".

3-12-2-2. Setting the FEM Coil Condition on a Periodic Boundary

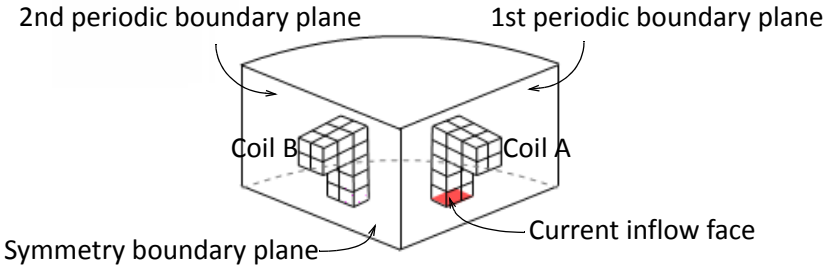
Set one FEM Coil condition for the coil connected through the periodic boundaries.

For example, the model shown in Example 1 has Coil A and Coil B that are connected through the periodic boundaries. For this model, select both Coil A and Coil B as the target region of the FEM Coil condition. Either Coil A or Coil B can be set as the inflow face.



Example 1

Example 2 shows a model that is half the model in Example 1 cut lengthwise.

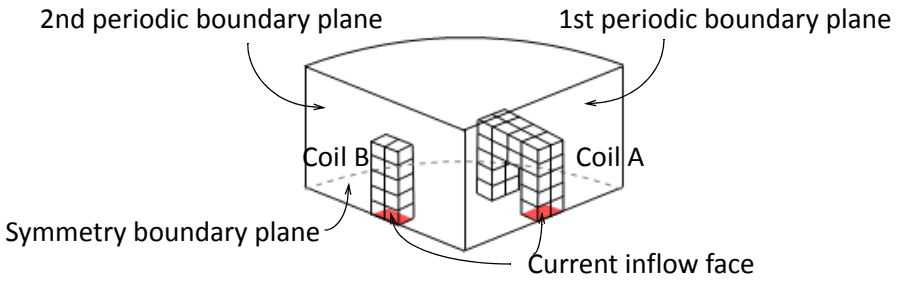


Example 2

Set one FEM Coil condition for the coil is connected through the symmetry boundary and periodic boundary as shown in Example 2. Be sure to set the element faces (highlighted in Example 2) on the symmetry boundary as the inflow face.

MEMO Either Coil A or Coil B can be set as the inflow face.

If the coil is divided, set both sections of the coil as the inflow face as shown in Example 3.



Example 3

3-13. Group (FEM Coil)

▶▶ Reference

- For more information, see "Group (FEM Coil)" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "FEM Coil" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-13-1. Function

By setting the FEM Coil Group condition, you can set multiple FEM coil conditions and manage them as one group. It can be used when setting the macro components, Lap Winding (Brush Motor) and Wave Winding (Brush Motor), in the circuit.

■ Related items

- Lap Winding (Brush Motor), Wave Winding (Brush Motor)
- FEM Coil condition

■ Settings

- FEM Coil condition

■ Available Modules

- 2D analysis, 3D analysis

3-13-2. Important Notes Regarding the Use of This Function

- This condition is essentially the same feature as the FEM Coil condition in terms of the type of condition it deals with.

3-14. FEM Conductor

▶▶ Reference

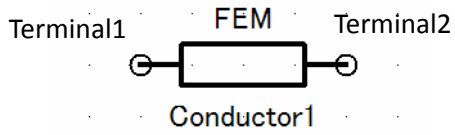
- For more information, see “FEM Conductor” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “FEM Conductor” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-14-1. Function

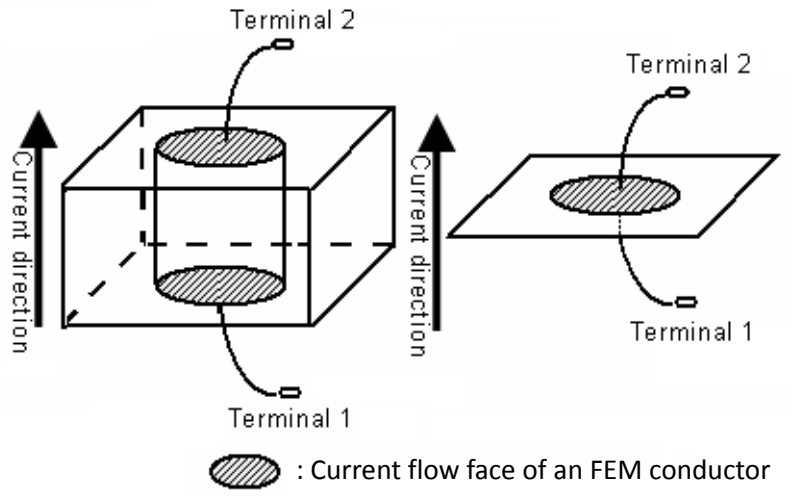
The FEM conductor condition is used to obtain the current distribution in a conductor by linking a model to a circuit. With this condition, the effect of eddy currents are taken into account.

In a 3D analysis, the inflow face and outflow face of the conductor need to be specified, and terminals are connected to each face.

In a 2D analysis, the face where the current flows needs to be specified, and terminals are connected to the front and back of the face as shown in the figure below



FEM conductor component (JMAG-Designer)



FEM conductor (3D) FEM conductor (2D)

■ Related items

- Material properties: Electrical properties

■ Related items

- Group (FEM Conductor) (140 page)
- External Circuit (147 page) *Mandatory
- Full Model Conversion (283 page)
- Circuit Conversion (286 page)

■ Settings

- Current flow face (2D analysis, axial symmetry analysis)
- Current inflow/outflow faces (3D analysis)
- FEM conductor condition to link

■ Available modules

- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

■ Output items

- Current density (A/m^2)
- Current in a circuit component (A)
- Electric potential at a terminal in a circuit (V)

3-14-2. Important Notes Regarding the Use of This Function

- A circuit needs to be created when an FEM conductor condition is set.
- One FEM conductor condition corresponds to one circuit component.
- Multiple FEM conductor conditions can be set for one model.

- The eddy currents occur in the conductor when an FEM conductor condition is set. The mesh modeling the skin effect needs to be generated.

  ▶▶ Reference

  - For information about the mesh generation, see Chapter 5, “Generating Mesh” in the JMAG-Studio User's Manual.
  - For information about the mesh generation, see Chapter 36 “Generating Mesh (Automatic)” in the JMAG-Designer User's Manual.

- A current flows through the inflow face, conductor, and outflow face when an FEM conductor condition is set. The electric conductivity must be set for the conductor. The inflow and outflow faces should not be set for parts other than conductor.

  ▶▶ Reference

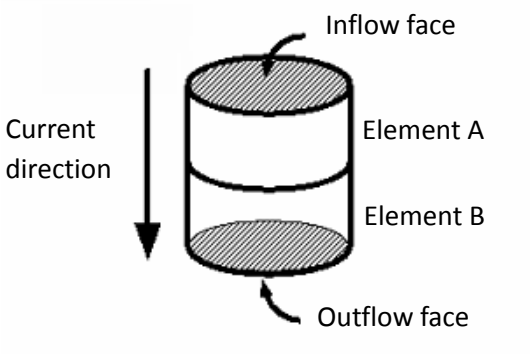
  For information about setting electrical conductivity, see P.31 "2-4-1. Specifying Electrical Conductivity/Electric Resistivity (Magnetic Field Analysis)".

- An FEM conductor condition and the allowing displacement current feature cannot be used at the same time.
- An FEM conductor condition cannot be set for the conductor that is divided by an antiperiodic boundary.
- The current value at the first step must be set to 0 A when an FEM conductor condition is used to take into account eddy currents in an analysis.
- Insulation can be set on a FEM conductor in a 2D analysis.
- The insulation cannot be set for an inflow face of a FEM conductor and a face that it is connected with for a 3D analysis.

  ▶▶ Reference

  - For more information about using insulation in material units, see "3-6-2. Allowing Electric Insulation" on page 99.
  - For more information about using the insulation condition, see "3-34. Insulation" on page 264.

- In a 2D analysis, the current flow region needs to be specified.
- In a 3D analysis, the inflow and outflow faces of the current need to be specified.



Group definition

  - The inflow and outflow faces need to be specified for the outer surface of a model.
  - The inflow and outflow faces need to be specified on periodic boundary faces if a conductor is divided by a periodic boundary.
  - A symmetry boundary condition needs to be specified on the inflow and outflow faces.
- An FEM condition can be set for multiple regions that are separated and for one region where multiple materials are assigned. The following are the examples of when this condition can and cannot be used.

■ Available

When more than one material properties are set on an area where an FEM conductor condition is set (Fig. 1).

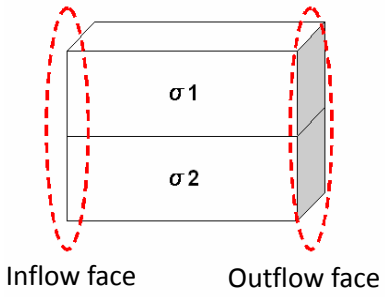


Fig. 1. Multiple material properties are set on an area where an FEM conductor condition is set (σ1, σ2: electric conductivity)

■ **Available**

When an FEM conductor condition is set for multiple regions that are separated (Fig. 2).

In this case, the faces that are set as an inflow face and an outflow face will be calculated individually so that the faces have the same voltage within each of the inflow and outflow faces.

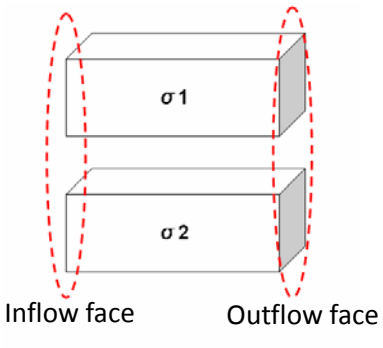


Fig. 2. One FEM conductor condition is set on discontinuous multiple regions (σ1, σ2: electric conductivity)

■ **Not available**

When the inflow faces or outflow faces set for multiple FEM conductor conditions are in contact with each other (Fig. 3).

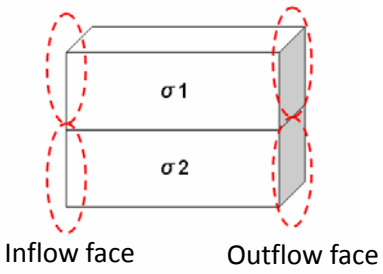


Fig. 3. Inflow faces or outflow faces set for multiple FEM conductor conditions are in contact (σ1, σ2: electric conductivity)

- In a 2D analysis, the height of an FEM conductor (in the Z-axis direction) can be specified using the full model conversion feature. If the height is not specified, the height of the FEM conductor is assumed to be the same as the model thickness.

3-15. Group (FEM Conductor)

▶▶ Reference

- For more information, see "Group (FEM Conductor)" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "FEM Conductor" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-15-1. Function

The FEM Conductor Group condition is the feature for setting multiple FEM conductor conditions at a time and managing them as a group. It can be referenced from the cage component in the coupled analysis with the circuit.

■ Related items

- Cage component
- FEM conductor condition

■ Settings

- FEM conductor condition

■ Available modules

- 2D analysis, 3D analysis

3-15-2. Important Notes Regarding the Use of This Function

- This condition is essentially the same feature as the FEM Conductor condition in terms of the type of condition it deals with.

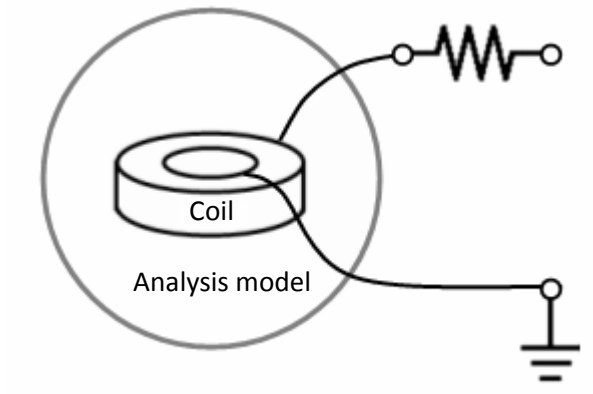
3-16. External Circuit

▶▶ Reference

- For more information, see Chapter 8 “Creating an External Circuit” in the JMAG-Studio User's Manual.
- For more information, see Chapter 35“Circuit” in the JMAG-Designer User's Manual.

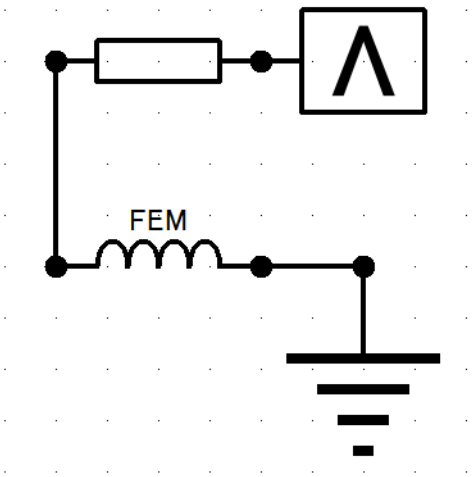
3-16-1. Function

The circuit is represented by placing the circuit components and connecting them with lines. Specify the voltage of the terminals in the circuit and current inflow for the circuit analysis run at the same time with the magnetic field analysis.



External circuit connection

When connecting the resistance component to the coil modeled as above, create a circuit as shown below.



Example of circuit setting

■ Related items

- FEM Coil (131 page)
- Group (FEM Coil) (140 page)
- FEM Conductor (141 page)
- Group (FEM Conductor) (146 page)
- Circuit Conversion (286 page)
- Power Source Component Control (JMAG-Designer) (293 page)
- User subroutine, *usrcir* (293 page) * JMAG-Studio Only
- Coupled analysis (magnetic field and thermal)

■ Settings

- Creating the circuit (arranging components and connecting terminals with wire)
- Creating component characteristics and setting each component
  - FEM Coil Component
  - FEM Conductor Component
  - Reference FEM Coil Component / Reference FEM Conductor Component
  - Resistor Component
  - Capacitor Component
  - Inductor Component
  - Switch Component
  - User Subroutine Component *Supported only for JMAG-Studio
  - Diode Component
  - Electric Potential Source (One Terminal) Component
  - Electric Potential Source (Two Terminals) Component
  - Current Source Component
  - Power Source Component
  - Brush/ Commutator Component
  - Three-phase Electric Potential Source Component/Three-phase Current Source Component
  - Macro Component
  - Electric Potential Probe Component
  - Current Probe Component
  - Ground Component

■ Available modules

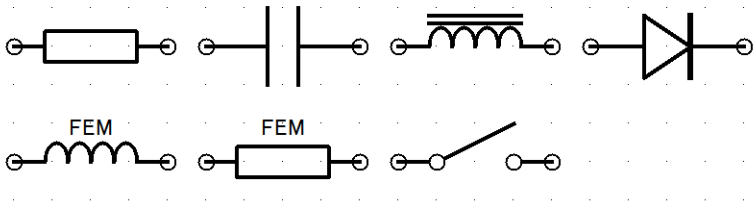
- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

■ Output items

- Voltage at circuit terminals (V)
- Current in a circuit component (A)
- Switching the components on/off
  = 0: Off
  = 1: On
  = -1: Electrically active (no switching)

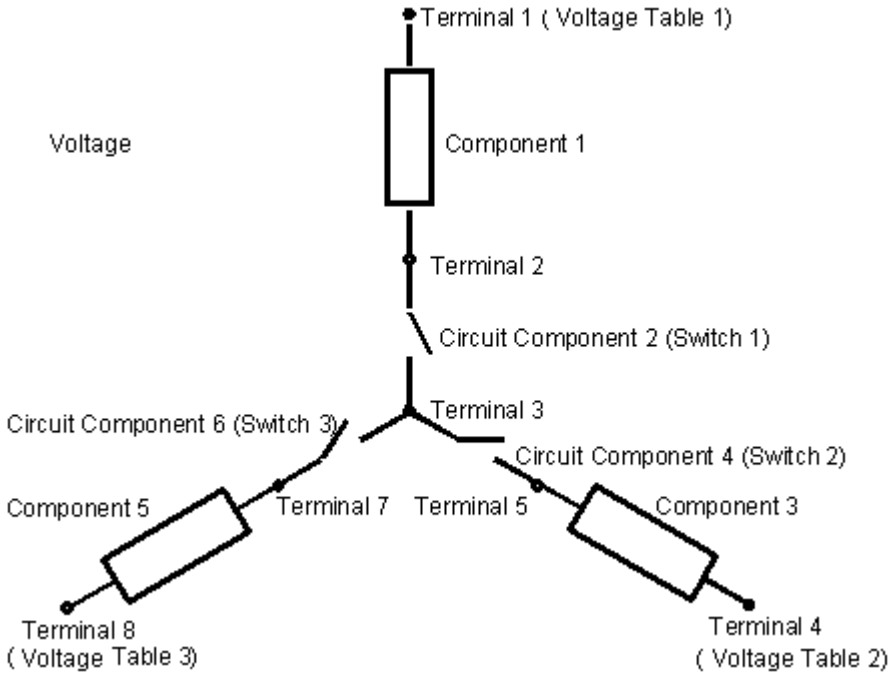
3-16-2. Important Notes Regarding the Use of This Function

- Specify the connection of circuit components.
- The components making up a circuit are connected together through terminals.
- To run the circuit analysis, the circuit component type needs to be specified for each component in the circuit after creating the circuit diagram.
- When the circuit components are arranged as below, the result is output assuming that the direction of current flowing from left to right is the positive direction.



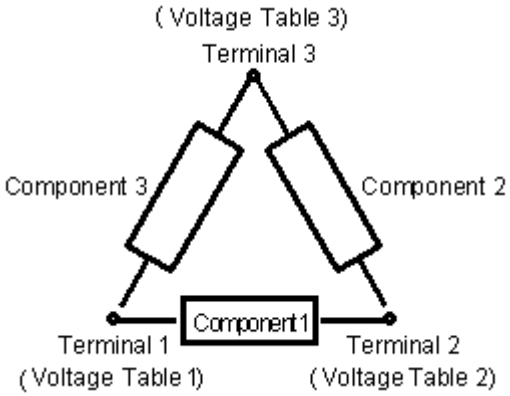
- The terminals located on both ends of all the circuit components making up the circuit need to be connected to the electric potential source component terminal at one or more points.

  For example, in the following diagram, analysis cannot be run if switches 1, 2, and 3 are all off at a certain moment. Make sure one of the switches is always on.



Example of Y-shaped connection

In the case of the following diagram, the analysis cannot be run if the Voltage for all of the terminals 1, 2, and 3 is not determined. Set the voltage of 0 V to one of the terminals to set the reference voltage.



Example of delta-shaped connection

3-16-3. FEM Coil Component

- FEM coil components need to be linked to FEM coil conditions.

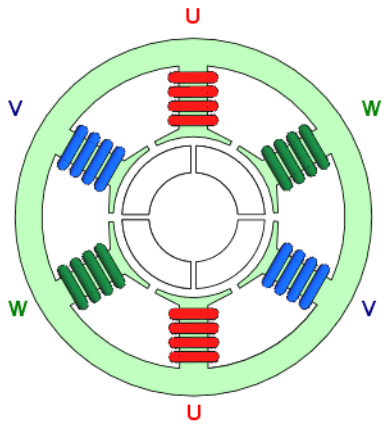
> **Reference**
> For more information about the FEM coils, see "3-12. FEM Coil" on page 131.

■ Specifying the number of turns

- The number of turns must be a positive real number.
- The number of turns cannot be applied to the circuit conversion. Specify half the value for the number of turns when the area of cross section for FEM coil is half when using a partial model.

> **Reference**
> For more information, see "3-42. Circuit Conversion" on page 286.

- Specify the number of wires wound around one slot.
  For example, the number of turns is 4 for the following three-phase concentrated winding motor.



Three-phase motor with 6-slot

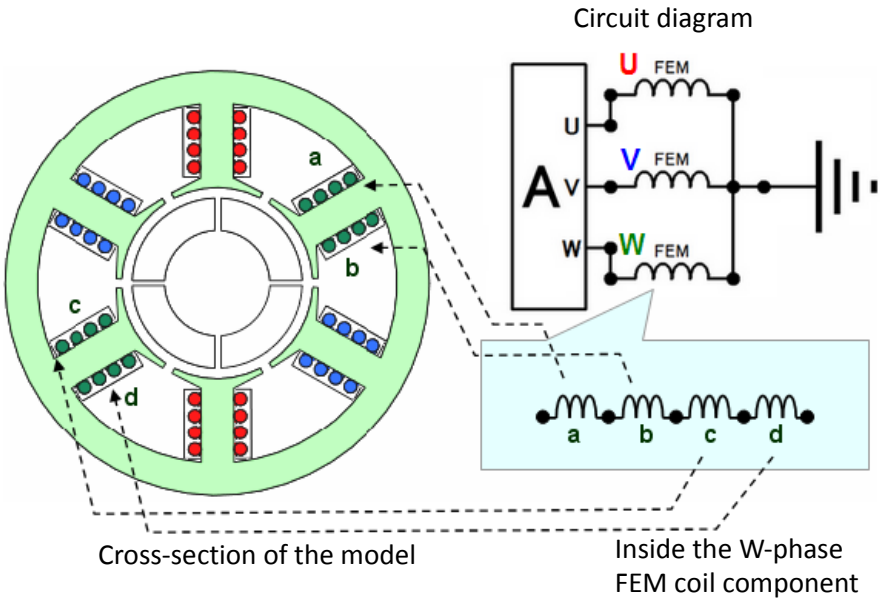
To perform a 2D analysis using this motor, the FEM coils need to be set as follows:

1. **Set an FEM coil condition for each phase.**
2. **Specify the coil regions of each phase (a, b, c, and d in the figure below) for each FEM coil condition.**

   The coil regions specified for each FEM coil condition are connected in series.

3. Arrange an FEM coil component for each phase in the circuit, and then link to each FEM coil condition.

With this setting, four coil regions are connected in series in one FEM coil component. Therefore, the number of turns is 4 for each FEM coil component.



■ Specifying the resistance

The resistance can be specified with the following types.

■ Constant

The constant is used only when the resistance value is constant.

■ Temperature Dependent (magnetic field analysis only)

- The temperature dependent is used for magnetic field analysis to evaluate the effects of the resistance that varies with temperature. The temperature dependence of the resistance and the temperature variation (time-dependent) can be set. When the values to be referenced are outside the specified range, the following methods are used.
  - Linearly extrapolates the temperature dependence of the resistance.
  - Applies the temperature defined for each end of the point sequence as a constant to the temperature variation (time-dependent).

■ Temperature Dependent (coupled analysis)

- The temperature dependent is used for referencing the temperature from the thermal analysis results.
- To enable this option, the average temperature calculation condition needs to be set in the thermal analysis.
- The temperature distribution in the coil is uniform because the temperature to be referenced is calculated with the average temperature calculation condition in thermal analysis.
- To take into account the temperature distribution in a coil, the FEM conductor component needs be arranged in the circuit.

3-16-4. FEM Conductor Component

- Specify the FEM conductor condition that is associated to the FEM conductor.

3-16-5. Reference Coil Component/Reference Conductor Component

- The reference conductor component is used when a partial model can be created for the FEM model, but not for the circuit model. With this component, the size of the analysis model can be reduced by creating a partial model of the FEM model, even if its circuit model is a full model.

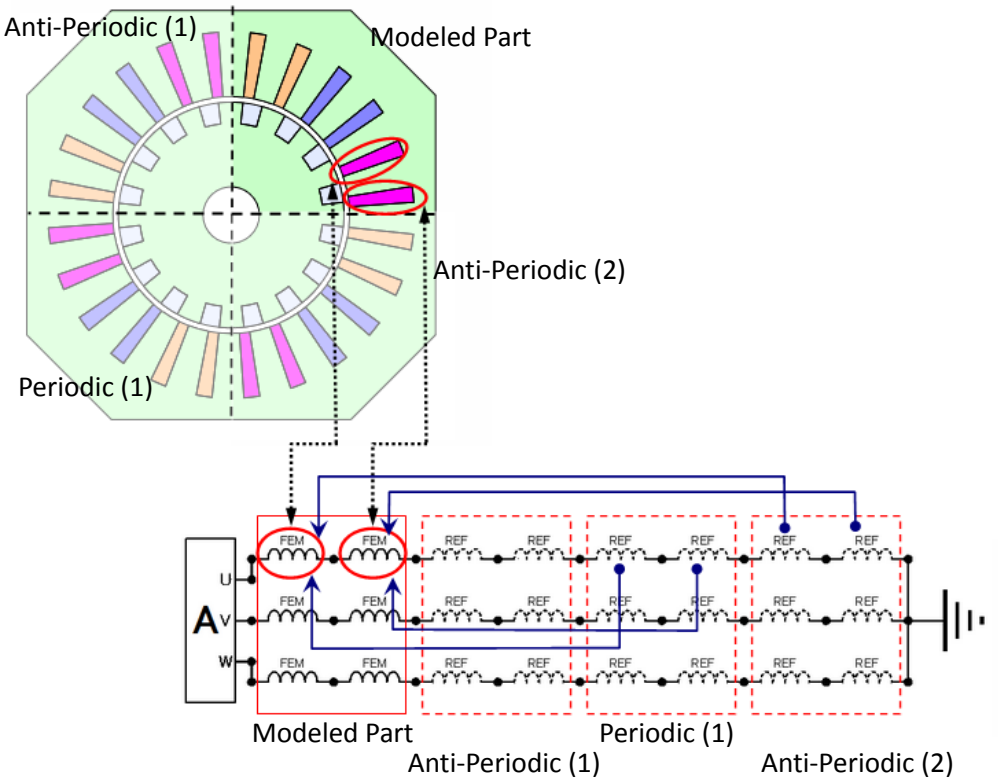
  The FEM coil and FEM conductor are linked with the FEM coil condition and the FEM conductor condition set in the full model respectively in the circuit. The reference coil and reference conductor components link with the predefined FEM coil and FEM conductor components respectively.

  In the analysis, the voltage at each end of the reference coil (conductor) component is solved so that it becomes the same value as the reference FEM coil (conductor). Therefore, this option can only be used for circuit models where the voltage at each end of the reference coil (conductor) component and that of the referenced FEM coil (conductor) are the same. Otherwise, the calculation cannot be run correctly. For example, when modeling a 2D induction machine, the circuit model has to include all the bars as components as a full model whereas an FEM model can be made into a partial model.
- The circuit conversion function cannot be used when the reference coil component or the reference conductor component is used. The conversion for the results needs to be set separately.

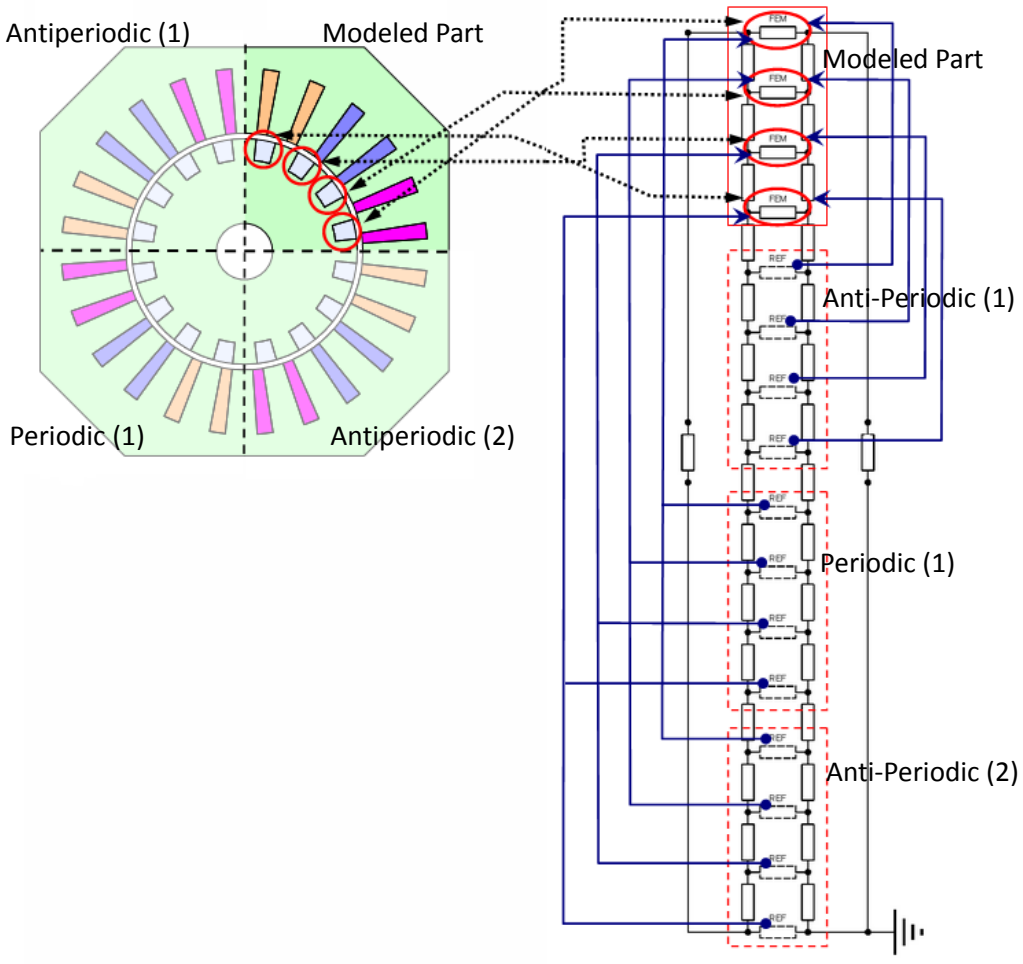
- The reference coil component cannot be used when the direct method (FEM+BEM) is used as the calculation method. The reference conductor component can be used regardless of the type of calculation method.
- Specify [Reference FEM Coil] (or [Reference FEM Conductor]).
- Set [Periodicity]. When a partial model is created for the FEM model, and the circuit is a full model, the sum of the FEM coil (or conductors) and reference coils need to match the number of model divisions. For the model with a periodicity, each reference coil (conductor) needs to be set to have either periodicity or antiperiodicity to match the setting on the partial model.

  Selecting [Periodic] sets the reference coil (conductor) component to be aligned with the FEM coil (conductor), whereas selecting [Anti-Periodic] sets it to the reverse direction, meaning that the current inflow reverses in the FEM model. The part of the model having a periodicity means that the reference coils (conductors) and FEM coil (conductors) have the current flowing in the same direction.

  The part having an antiperiodicity means that the reference coils (conductors) and FEM coil (conductors) have the current flowing in the opposite directions.



The figure above shows an induction motor and its coils. A 1/4 model is created and the Antiperiodic Boundary condition is set. For creating the circuit for the full model, the reference coils are used to represent the rest of the model (anti-periodic(1), periodic(1), antiperiodic(3) in the figure). All these reference coils reference the FEM coil in the partial model, however, the FEM coil used in antiperiodic(1) and (2) is set as Antiperiodic, and for periodic (1) Periodic is set. Because of this, the components in antiperiodic (1) and (2) are aligned in the direction opposite to the FEM coil. Despite that the current flows in the same direction in both the FEM coil and reference coils on the circuit, in antiperiodic (1) and (2), the current direction is reversed due to the periodicity when reflected on the partial model. As a result, the flux density distribution generated has an antiperiodicity. Thus, the reference coils need to be set to periodic or antiperiodic in accordance with the periodicity of the part they represent. (whether the model part has a periodicity or antiperiodicity needs to be taken into account when setting the reference coils to periodic or antiperiodic.)



The figure above shows the same induction motor and its conductors. The reference conductors are arranged so as to reference the FEM conductor in the partial model. As in the case of the reference coils, the rest of the model consists of antiperiodic (1), (2), and periodic (1). The reference conductors for antiperiodic (1), and (2) are set as Antiperiodic, and periodic (1) is set as Periodic. Just like the coils, even if the current flows in the same direction in the reference conductors and FEM conductor that are set as Antiperiodic, the reference coil component set as Antiperiodic is aligned in the opposite direction and therefore the current flows in the reverse direction in the FEM model.

3-16-6. Resistor Component

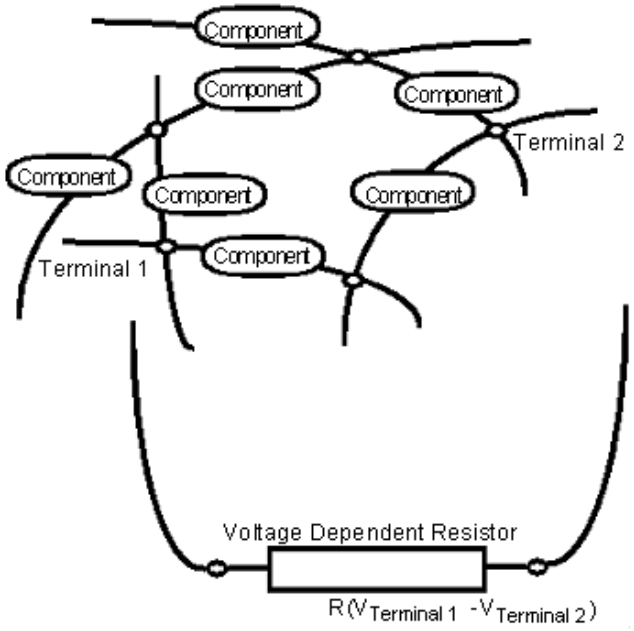
- If selecting constant for the resistance type, enter a positive number for the resistance value.
- Specifies resistance value using point sequence when either [Time], [Angle], [Distance], [X-axis Direction], [Y-axis Direction], [Z-axis Direction], or [Step] is selected as a resistance type.
  Each point sequence for [Angle], [Distance], [X-axis Direction], [Y-axis Direction], and [Z-axis Direction] are used periodically.

  MEMO [X-axis Direction], [Y-axis Direction], and [Z-axis Direction] are only available in JMAG-Studio. Select [Distance] for JMAG-Designer.
- Specify the following items when [Voltage] is selected as the resistance type.
  - Resistance for the first analysis step (initial status)
  - 1st referenced electric potential probe and 2nd referenced electric potential probe for JMAG-Designer
  - Referred terminal 1 and referred terminal 2 for JMAG-Studio.

- Voltage dependent resistance table

For electric voltage dependence, a circuit component can be specified based on the voltage potential difference between two designated points in the circuit (see the diagram below).

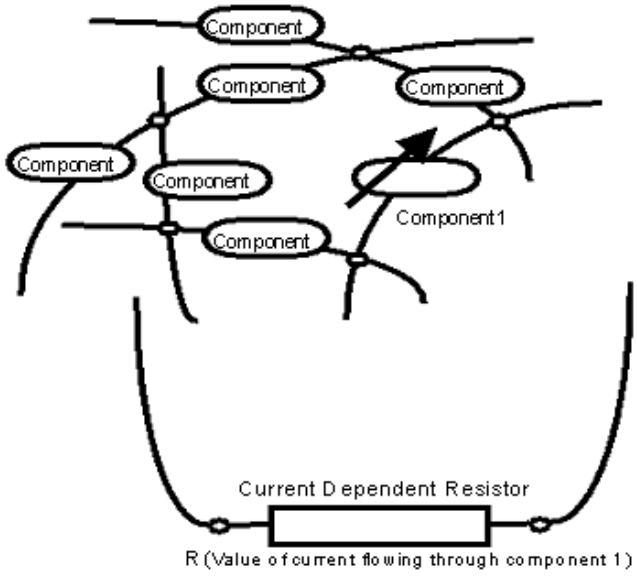
At the 1st step, calculation is run based on the specified initial status (resistance value). Thereafter, calculation is run by determining the status with the point sequence based on the voltage potential difference between the designated terminals.



Voltage dependent resistance

- Specify the following items when [Current] is selected as the resistance type.
  - Resistance for the first analysis step (initial status)
  - Reference component
  - Current dependent resistance table

  For current dependence, the status of the circuit component can be specified based on the current flowing in the specified circuit component in the circuit (see the diagram below). At the 1st step, calculation is run based on the specified initial status (resistance value). Thereafter, calculation is run by determining the status from the point sequence based on the current flowing in the designated circuit components.



Current dependent resistance

- The resistance can be specified with the following types.
  - Temperature Dependent (magnetic field analysis only)
    - This is used for magnetic field analysis to evaluate the effects when the resistance varies with temperature.
    - The temperature dependence of the resistance and temperature variation (time-dependent) need to be set.
    - When the values to be referenced are outside the specified range, the following methods are used.
      - Linearly extrapolates temperature dependence of the resistance.
      - Applies the temperature defined for each end of the point sequence as a constant to the temperature variation (time-dependent).

- Temperature dependence for coupled analysis
  - This is used for referencing the temperature from the thermal analysis results.
  - To enable this option, the average temperature calculation condition needs to be set in the thermal analysis.

MEMO If setting the temperature dependence of resistance, the setting is not applied to the component itself but to the component type. Therefore, in order to evaluate the effects of the temperature on each component, you need to set another component type.

3-16-7. Capacitor Component

- Specify the capacitance by entering a positive value.

3-16-8. Inductor Component

- Specify the inductance by entering a positive value.

3-16-9. Switch Component

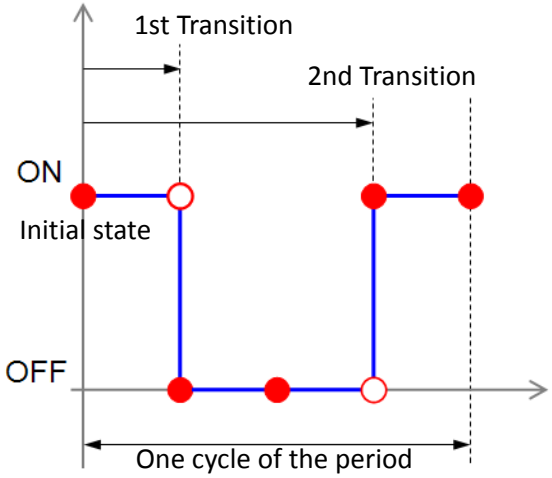
- The switch component cannot be used in a frequency response analysis.
- If selecting time dependence, rotation angle dependence, distance dependence, X-axis direction displacement dependence, Y-axis direction displacement dependence, Z-axis direction displacement dependence, or step number dependence for the switch type, specify the timing of switching on/off. The following two methods are available for specifying the timing.

  MEMO [X-axis Direction], [Y-axis Direction], and [Z-axis Direction] are only available in JMAG-Studio. Select [Distance] for JMAG-Designer.

■ Parameters

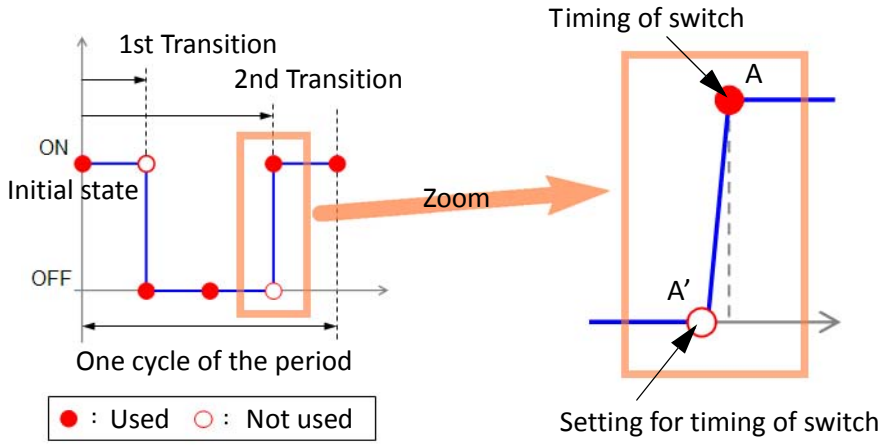
When the switch timing has a periodicity, specify the following items.

- One cycle of the period
- 1st Transition
- 2nd Transition
- Initial status (ON/OFF)



Parameter specification

Specify the settings in small increments from the step that is reflected by the timing of the switch when specifying the time dependency for the switch.



Time dependency switch

In this example, the time for A' is specified as the [Second Transition] in order to time the switch to A. This setting also applies for [First Transition].

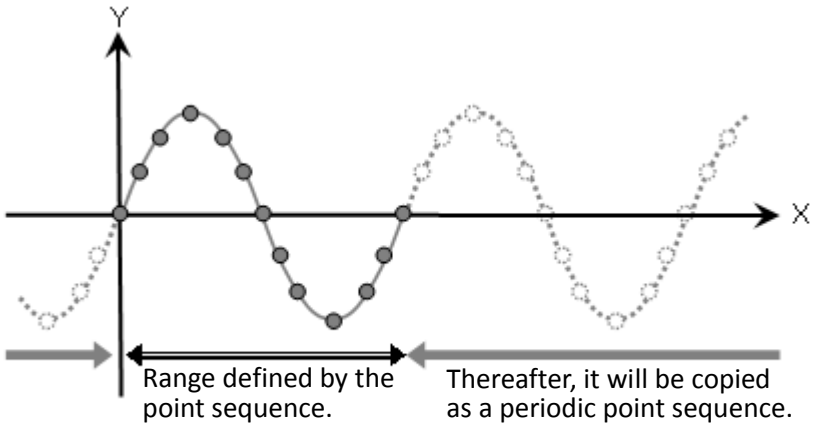
■ Point sequence

When the switching timing is complicated, specify the timing by creating a table.

- When it is 1, the switch is on, and when it is zero, it is off.
- Make sure to define more than two points {(X1,Y1),(X2,Y2)} to create a point sequence. X1 and X2 need to meet the condition: X1<Xmin<X2 and X1<Xmax<X2, where the Xmin is the minimum value of the X-axis, and Xmax, the maximum value, to be referenced in the analysis.
- The pulse can be represented by a table.

▶▶ Reference
For example of creating the table, see P.120 "3-10-2-2. Point Sequence Data".

- Selecting [Periodic Point Sequence] allows the point sequence to be copied in a periodic sequence and used in the analysis if the X-axis value exceeds the range defined by the point sequence during the calculation.



If the [Periodic Point Sequence] check box is selected:

When the [Periodic Point Sequence] check box is cleared, in case an X-axis value that exceeds the range defined by the point sequence data is referred to during the calculation, a warning will be output to the solver report.

- Specify the following items when [Voltage] is selected as the switch type.
  - Initial switching condition (first analysis step)
  - 1st referenced electric potential probe and 2nd referenced electric potential probe for JMAG-Designer
  - Referred terminal 1 and referred terminal 2 for JMAG-Studio

- Voltage dependent switch table

In the case of the electric voltage dependence, it is available by designating arbitrarily two points in the circuit to set the status of the circuit component by using the voltage difference between the two points.

At the 1st step, calculation is performed based on the initial status of open/close that was set. At the 2nd step or later, calculation is performed by determining the status from the timing specified on the basis of the voltage difference generated between the designated terminals.

- Specify the following items when [Current] is selected as the switch type.
  - Initial switching condition (switching condition first analysis step)
  - Reference component
  - Current dependent switch table

In the case of the current dependence it is available by designating an arbitrary circuit component in the circuit, to set the status of the circuit component based on the current value flowing in the circuit component.

At the 1st step, calculation is performed based on the initial status of open/close that was set. At the 2nd step or later, calculation is performed by determining the status from the timing specified on the basis of the value of the current that flowed in the circuit.

3-16-10. User Subroutine Component (JMAG-Studio)

- Specify the identifier within the user subroutine *usrcir*. Input the negative integer in this item.

3-16-11. Diode Component

- In the diode component, the flow from the 1st to 2nd terminal is supposed to be in the forward direction.
- The "Referenced Initial Voltage" is used to define which point on the current-voltage characteristics of the diode is to be used in the first nonlinear calculation at the 1st step. The default value is 0 V.

  In the case where nonlinear iteration converges, this setting will not affect the analysis results. When the operating voltage is known, by setting such a voltage as the initial voltage beforehand, there are situations when the number of nonlinear iterations can be reduced.

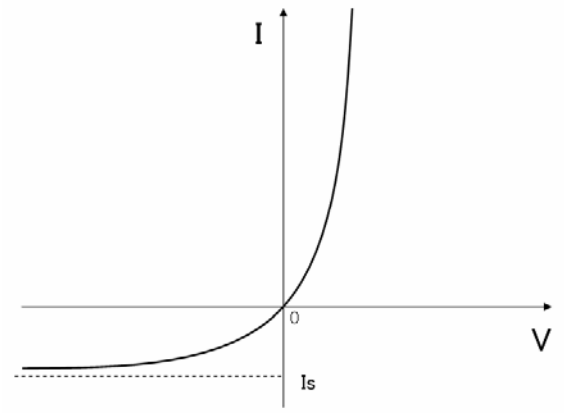
- The diode component varies impedance with the current-voltage characteristic curve in nonlinear iterative calculation. You should set the upper and lower limits of the impedance.

  Note that setting the value extremely small or large could increase the number of non-linear iterative calculations. For example, if the operating voltage can be expected, set the upper and lower limits by estimating the impedance at the voltage.
- The specification method for the electric properties of the diode components can be chosen from the following two ways.

■ Specifying method using the logarithmic function

$$V(t) = K\ln\left(\frac{I_s + I(t)}{I_s}\right)$$

Where, I_s is the saturated current (A), $V(t)$ is the terminal potential across a diode (V), $I(t)$ is the current flowing through the diode (A), K is the parameter, and ln is the natural logarithmic function (See figure below).



Electrical Property of a Diode Defined by the Logarithmic Function

The value to be entered for the saturation current Is should be of the order of μA.

■ Method of specifying, using the point sequence, the relationship between voltage difference of the terminals at both ends of the diode and current value

- In the case of having specified by using the point sequence, the calculation may have bad convergence. In such a case, perform approximation calculation using the logarithmic function or re-examine the point sequence.

  There may be, depending upon the point sequence, the case of the difficult convergence of a nonlinear iterative calculation because the nonlinear property of the diode drastically changes its impedance value. In the case of having specified by using the logarithmic function, however, convergence of the iterative calculation will not be as bad as that specified using the point sequence.
- When generating the point sequence, it is required to decrease the variation amount of impedance among the points by increasing their number and enhancing their expressiveness, or to express in a simplified manner by reducing the number of the unnecessary points.
- To set the properties of the diode component using a point sequence, the voltage should be defined from the negative to positive voltage ranges as shown below.

Voltage (V)	Current (A)
-10000	0
0	0
10000	100

In the above example, if the voltage is applied in the forward direction, it works in the same way as a 100 ohm resistor component, if the voltage is applied in the reverse direction, it shows infinite resistance. In practice, the upper limit of impedance is set, and so calculation is performed using the upper limit.

In addition, create the point sequence to include the origin (0, 0) as in this example.

3-16-12. Electric Potential Source (1 Terminal) Component

- Specify either an electric potential source (one terminal) component or a ground component when using circuits.
- The time, rotation angle, translational distance, and step number can be treated as the X-axis of a function, and can also be specified by the user subroutine.
  In case the type of analysis is frequency response analysis (FQ), it is also possible to handle the frequency.
- It is possible to set offset to each function. At this time, the X-axis value and voltage value will shift the specified amount.
- The definition method of the voltage can be chosen from the following. Note, however in the case of frequency response magnetic field analysis (FQ), that the choice should be made out of three methods: constant value, sinusoidal function, or voltage point sequence.

■ Constant value (TR, FQ, DP)

This is used when the voltage does not change during the calculation, or when the voltage is changed by an external program when using dll.

MEMO In the frequency response analysis, constant means that the voltage does not vary with the frequency. Enter the amplitude in the text box. The phase is treated as zero.

■ Sinusoidal function (TR, DP, FQ)

The sinusoidal function is defined by amplitude *A*, frequency *F*, and phase *P*.

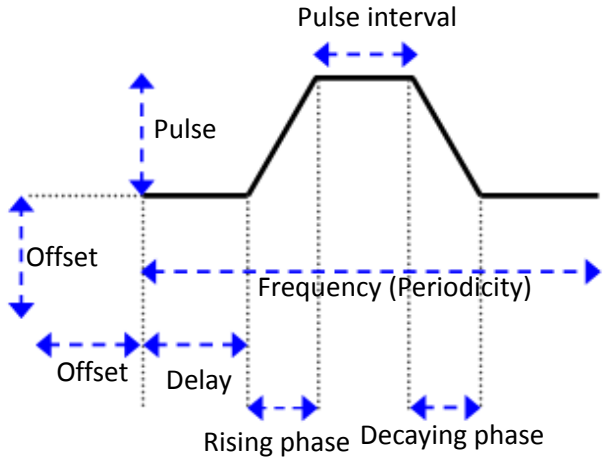
$$I(t) = A \cdot \sin(2\pi \cdot F + P \cdot \pi / 180)$$

In the frequency response analysis, frequency can not be specified in supply current condition.

■ Pulse function (TR, DP)

Define a pulse function by using the following values: pulse (A), delay (sec), pulse interval (sec), frequency (Hz), rising phase (sec), and decaying phase (sec).

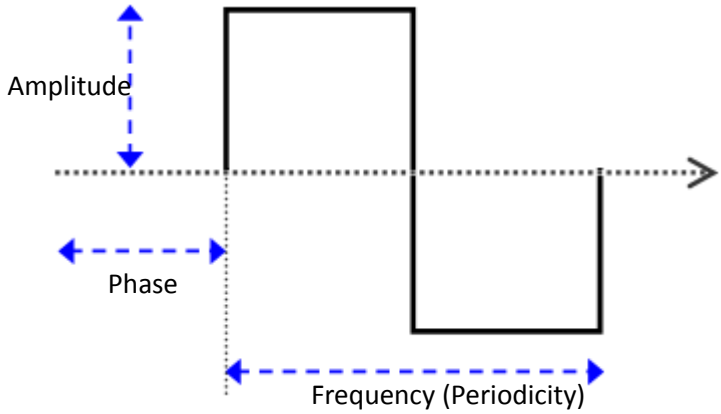
$$I(t) = C_0 + AMP \cdot \exp(-t/T)$$



When zero is specified at the [Rising phase] text box, the raised state is used. The same applies to the decaying case.

■ Rectangular wave function

Define a rectangular wave function by using amplitude (V), phase (deg), and frequency (Hz).



The switched state is used at the switching.

■ Exponential function (TR, DP)

The exponential function is defined by initial value C_0, amplitude *AMP*, and time constant *T*.

■ Voltage point sequence (TR, FQ, DP)

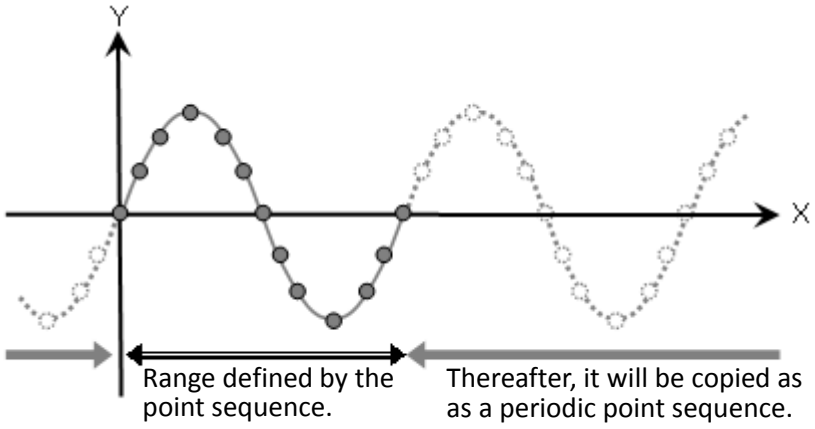
Define an voltage point sequence function by using a table of the time or other parameters, and the current value.

- For the voltage point sequence, you should enter two or more point sequences such as (X1, Y1), (X2, Y2). In this case, X1 and X2 must meet the conditions X1<Xmin<X2 and X1<Xmax<X2 regarding the minimum value Xmin and the maximum value Xmax of the X-axis parameters that are referred to in the analysis.
- The pulse can be represented by a table.

  ▶▶ Reference

  For example of creating the table, see P.120 "3-10-2-2. Point Sequence Data".

- When the [Periodic Point Sequence] check box is selected, in case the X-axis value exceeds the range defined by the point sequence data during the calculation, the point sequence will be copied periodically and be referred to.



If the [Periodic Point Sequence] check box is selected:

When the [Periodic Point Sequence] check box is cleared, in case an X-axis value that exceeds the range defined by the point sequence data is referred to during the calculation, a warning will be output to the solver report.

■ Coupled external circuit simulator (TR, DP)

This is selected when this supply voltage is used as the input terminal when conducting direct coupling with an external circuit simulator designated by JMAG. Furthermore, this function corresponds to only transient response analysis (TR and DP). In addition, at the time of use, it is necessary to set the coupling conditions of the external circuit simulator at the same time.

3-16-13. Current Source (1 Terminal) Component

- The current source component is used in combination with the electric potential source component.
- The time, rotation angle, translational distance, and step number can be treated as the X-axis of a function, and can also be specified by the user subroutine.
  In case the type of analysis is frequency response analysis (FQ), it is also possible to handle the frequency.
- It is possible to set offset to each function. At this time, the X-axis value and voltage value will shift the specified amount.
- The definition method of the current can be chosen from the following. Note, however in case of frequency response magnetic field analysis (FQ), that the choice should be made out of three methods: constant value, sinusoidal function, or current point sequence.

■ Constant value (TR, FQ, DP)

This is used when the current value does not change during the calculation, or when the current value is changed by an external program when using dll.

MEMO In the frequency response analysis, constant means that the current does not vary with the frequency. Enter the amplitude in the text box. The phase is treated as zero.

■ Sinusoidal function (TR, DP, FQ)

The sinusoidal function is defined by amplitude *A*, frequency *F*, and phase *P*.

$$I(t) = A \cdot \sin(2\pi \cdot F + P \cdot \pi / 180)$$

In the frequency response analysis, frequency can not be specified in supply current condition.

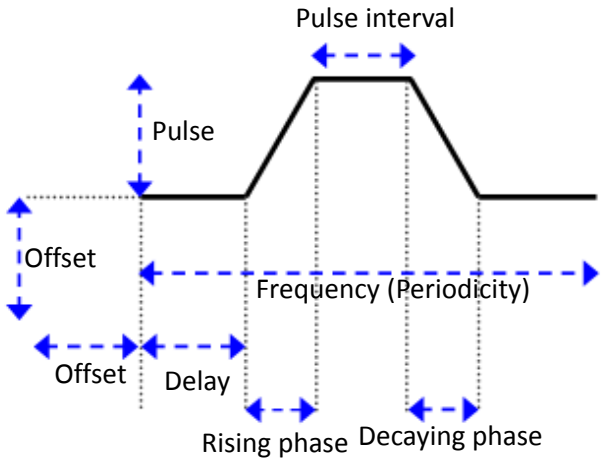
■ Exponential function (TR, DP)

The exponential function is defined by initial value C_0, amplitude *AMP*, and time constant *T*.

$$I(t) = C_0 + AMP \cdot \exp(-t/T)$$

■ Pulse function (TR, DP)

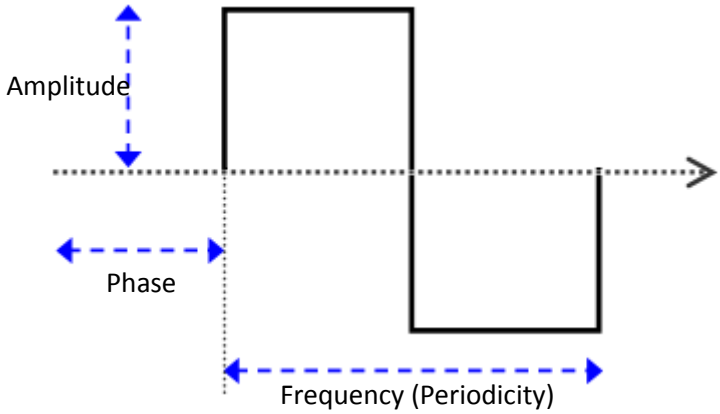
Define a pulse function by a pulse value (A), delay (sec), pulse width (sec), frequency (Hz), rise time (sec), and decay time (sec).



When zero is specified at the [Rising phase] text box, the raised state is used. The same applies to the decaying case.

■ Rectangular wave function

Define a rectangular wave function by amplitude (A), phase (deg), and frequency (Hz).



The switched state is used at the switching.

■ Harmonic (TR, DP)

Specifies distortion in waveform from a fundamental harmonics and higher harmonics.

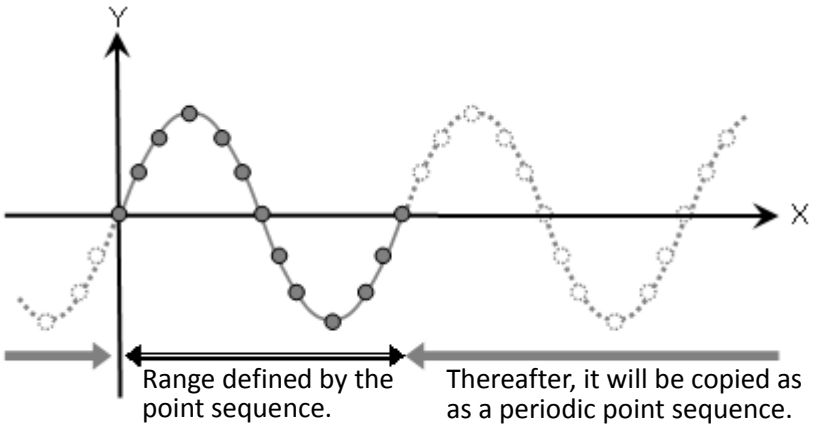
■ Current point sequence (TR, FQ, DP)

Define an current point sequence function by a table of the time or other parameters, and the current value.

- For the current point sequence, enter two or more points such as (X1, Y1), (X2, Y2). In this case, X1 and X2 must meet the conditions X1<Xmin<X2 and X1<Xmax<X2 regarding the minimum value Xmin and the maximum value Xmax of the X-axis parameters that are referred to in analysis.
- The pulse can be represented by a table.

  ▶▶ Reference

  For example of creating the table, see P.120 "3-10-2-2. Point Sequence Data".
- When the [Periodic Point Sequence] check box is selected, in case the X-axis value exceeds the range defined by the point sequence data during the calculation, the point sequence will be copied periodically and be referred to.



If the [Periodic Point Sequence] check box is selected:

When the [Periodic Point Sequence] check box is cleared, in case an X-axis value that exceeds the range defined by the point sequence data is referred to during the calculation, a warning will be output to the solver report.

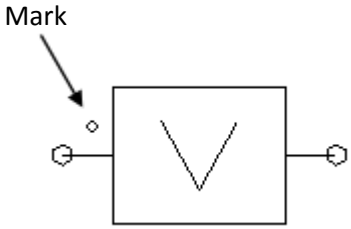
■ Coupled external circuit simulator (TR, DP)

This is selected when this supply current is used as the input terminal when conducting direct coupling with an external circuit simulator designated by JMAG. Furthermore, this function corresponds to only transient response analysis (TR and DP). In addition, at the time of use, it is necessary to set the coupling conditions of the external circuit simulator at the same time.

3-16-14. Electric Potential Source (2 terminals) Component

- Put at least one supply voltage or grand in the circuit network if you use this component.
- The time, rotation angle, translational distance, and step number can be treated as the X-axis of a function, and can also be specified by the user subroutine.
  In case the type of analysis is frequency response analysis (FQ), it is also possible to handle the frequency.
- It is possible to set offset to each function. At this time, the X-axis value and voltage value will shift the specified amount.
- The definition method of the voltage can be chosen from the following. The same function and point array for the supply voltage with one terminal can be applied.

  Note, however in the case of frequency response magnetic field analysis (FQ), that the choice should be made out of three methods: constant value, sinusoidal function, or voltage point sequence.
- For JMAG-Studio, using the marked terminal as reference, set the voltage difference to the other terminal.



3-16-15. Power Source Component (JMAG-Designer)

- The power source component can be used in a frequency response analysis (FQ).
- Only one power source component can be used at the same time with another.
- Use the power source component with the ground component.
- The power source component is used when running a two-way coupled magnetic field and thermal analysis for an electric induction motor.
- Constant or time dependent can be specified for electrical power.
- The time specified in the step control of a thermal analysis is referred from magnetic field analysis when using time dependent electrical power. Therefore, an electrical power for a time dependency that is within the time area specified in a step control of a thermal analysis needs to be specified. For example, when 0.0 seconds to 10.0 seconds is specified in a step control of a thermal analysis, specify the area of time for electrical power in a power source component to 0.0 seconds to 10.0 seconds.
- Specify the power source component control in a study property when using the power source component.

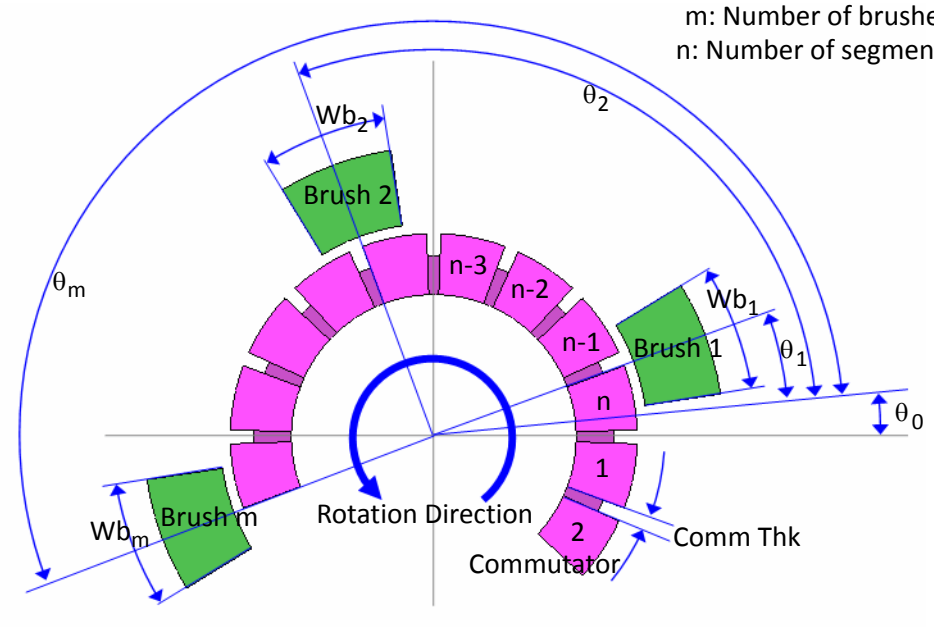
▶▶ Reference

- For more information about power source component, see "3-43. Power Source Component Control (JMAG-Designer)" on page 293.
- For more information about the two-way coupled magnetic field and thermal analysis, see "10-5. Coupling Magnetic Field Analysis and Thermal Analysis in Both Directions" on page 783.

3-16-16. Brush/Commutator Component

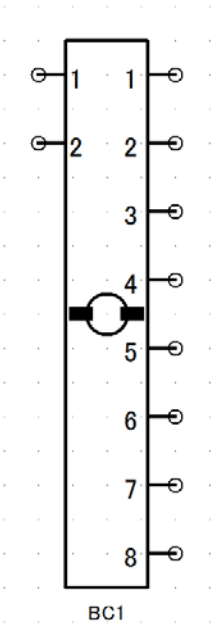
- The settings for a brush/commutator component are as follows:
  - Number of Divisions: Maximum value is 8.
  - Number of Brushes: Maximum value is 16.
  - Number of Segments
  - Slit Width between Segments (deg)
  - Brush Width (deg)
  - Initial Position of Brush (deg)
  - Contact Resistance between Brush and Commutator (ohm)
- The settings for an arc voltage as follows:
  - Arc Voltage (V)
  - Sparkover Voltage (V)

- The figure below shows a model with m brushes (from 3 to m-1 are omitted) and n segments. The segments are numbered clockwise. The brushes are numbered counter-clockwise. CommThk indicates the slit width between the segments.



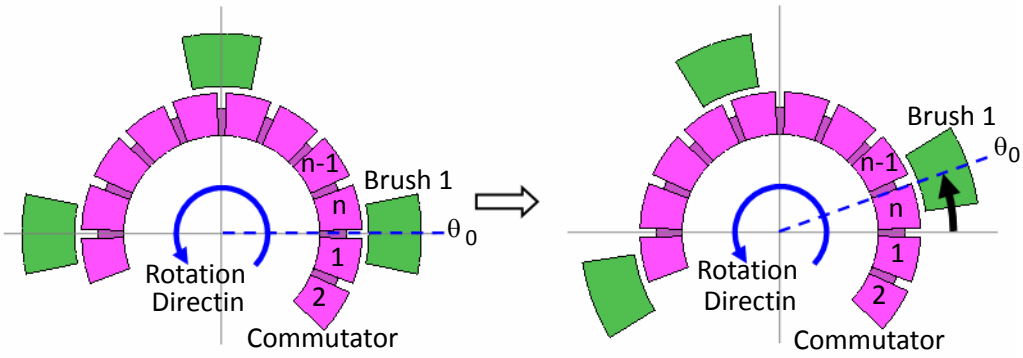
Brush and commutator

The created component should look similar to the figure below, with the brushes on the left side and the segments on the right side of the component.



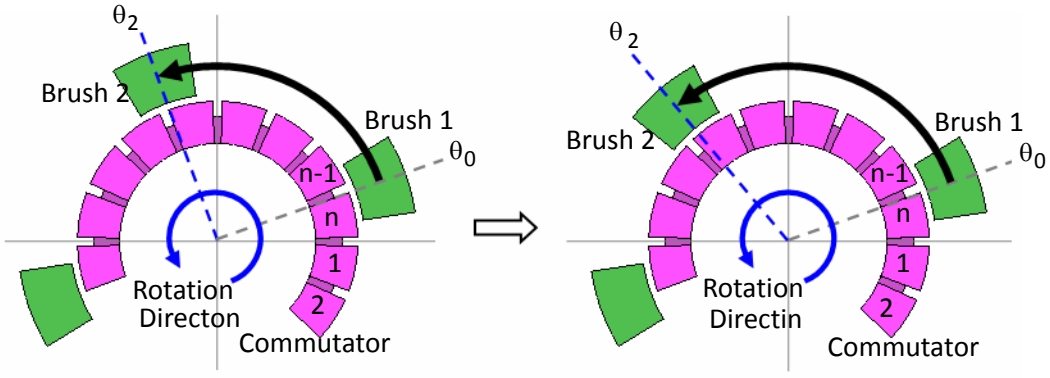
Brush/Commutator component

- The number of brushes and segments for a full model needs to be specified.
- The number of divisions needs to be specified when a partial model is created in the circumferential direction. The number of terminals of the brush/commutator component (on the segment side) varies depending on the number of divisions in a model. The number of brushes and segments needs to be multiplied by the number of divisions.
- The number of divisions needs to be specified by considering the periodicity of a model. If an antiperiodic model is used, the number of divisions needs to be divided by two. For example, the number of divisions is not 4 but 2 for a 1/4 model with an antiperiodic boundary.
- The initial position of the brush is where the center of brush 1 is positioned between segment 1 and n (θ_0 in the figure above). To displace the initial position to the rotation direction (counterclockwise) by 10 degrees, enter '10', and to displace it to the other direction (clockwise), enter '-10'.



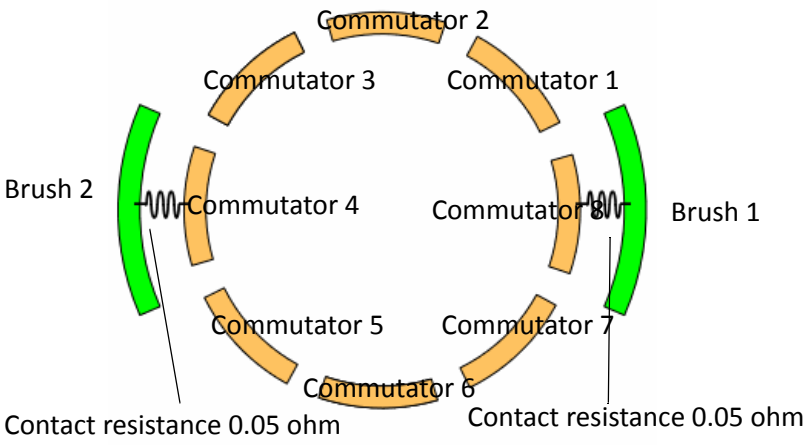
θ_0=0 θ_0=20

- The positions of brushes can be set individually (θ_1-θ_m) with respect to the initial position of the brush (θ_0).



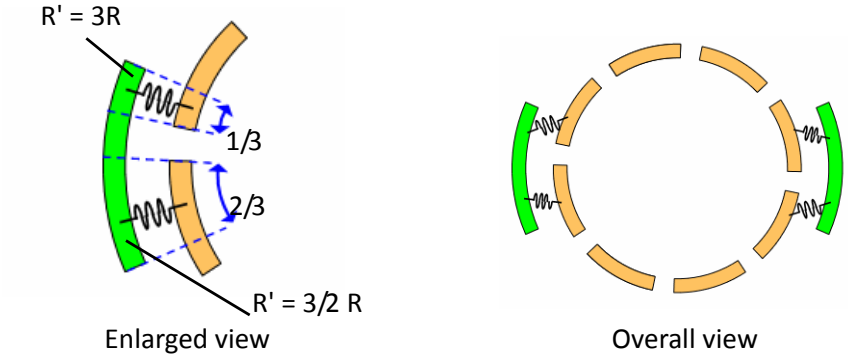
θ_0=20, θ_1=0, θ_2=90 θ_0=20, θ_1=0, θ_2=110

- The angle of the face where the brush and segment contact needs to be specified for the brush width. The brush width can be set individually (Wb_1-Wb_m in the figure above).
- The contact resistance when a brush and commutator contact completely must be specified. For example, the contact resistance is '0.05' in the figure below.



Brush and commutator contact completely

The contact resistance increases when a brush and commutator contact partially during an analysis. For example, the contact resistance is calculated based on the contact area when a brush and two commutators contact as shown in the figure below.



Brush and commutators contact partially

- The contact resistance between a brush and commutator in a rotating motor is expressed as the following equation (see the figure "Contact angle and contact resistance" below.)

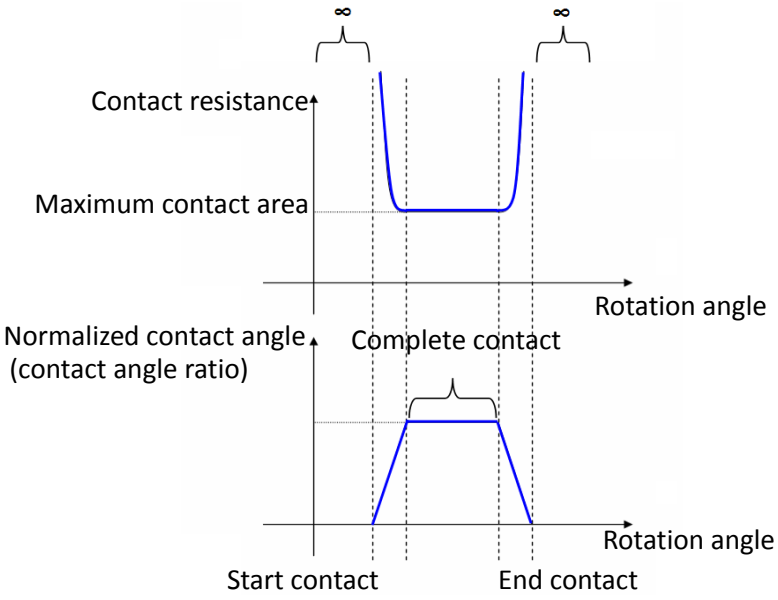
$$\text{Contact resistance between brush and commutator} = \frac{\text{Contact resistance for complete contact}}{\text{Contact angle}}$$

$$\text{Contact angle ratio} = \frac{\text{Contact angle}}{\text{Maximum contact angle}}$$

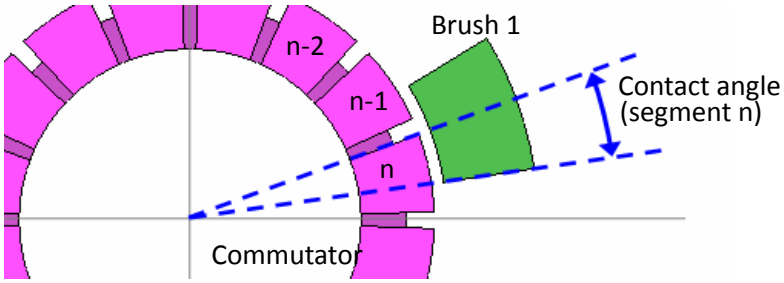
The maximum contact angle indicates the angle when a brush and commutator contact completely.

Maximum contact angle = Brush width (when brush width < commutator width)
Bar width (when brush width > commutator width)

For contact angle, see the figure "Contact angle" below.



Contact angle and contact resistance



Contact angle

- If the shape of a brush is not uniform in the rotation direction due to wear and the contact resistance does not symmetrically vary with each contact angle, the contact resistance can be determined based on the contact area ratio by specifying a contact area versus contact angle point sequence.

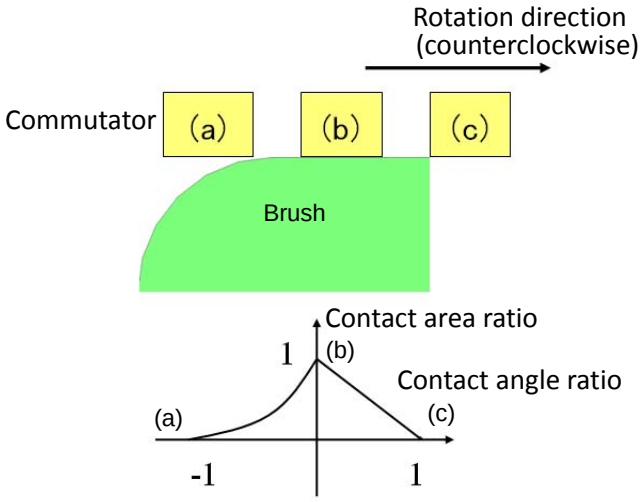
  The contact area ratio is obtained by normalizing the contact area by the maximum contact area. In this case, the contact resistance is expressed as following equation.

$$\text{Contact resistance} = \frac{\text{Contact resistance at the maximum contact area}}{\text{Contact area ratio}}$$

$$\text{Contact area ratio} = f(\text{Contact angle ratio})$$

  * 'f' is a function expressed as a point sequence

  In the (contact angle ratio, contact area ratio) point sequence, the contact angle has positive and negative values. The counterclockwise rotation is in the positive direction, and the contact area ratio =1 is the origin. For example, when a motor rotates counterclockwise, the brush is in contact with a commutator when the contact angle ratio = -1 and contact area ratio = 0, and it loses contact when the contact angle ratio =1 and contact area ratio = 0. To create a contact angle versus contact area point sequence, (contact angle ratio, contact area ratio) need to set using three points (-1,0), (0,1), and (1,0).



Point sequence of contact angle vs. contact area
Be sure to specify the status of (a), (b), and (c).

3-16-17. Three-phase Electric Potential Source Component / Three-phase Current Source Component

- In case the circuit component is a three-phase electric potential source or three-phase current source, define the following items.
  - X-axis Type
    - Time
    - Angle
    - Distance
    - X-direction
    - Y-direction
    - Z-direction
    - Step
  - Three-phase AC

    Choose from the following two types with regard to the phase difference among U, V, and W.
    - The type of -120 degree shift for V-phase and +120 degree shift for W-phase from U-phase as the standard.
    - The type of +120 degree shift for V-phase and -120 degree shift for W-phase from U-phase as the standard.
  - Amplitude
  - Frequency

    Specify this if the X-axis type is time dependence.
  - Period (mechanical angle)

    Specify this if the X-axis type is rotation angle dependency.
  - Periodic Distance

    Specify this if the X-axis type is distance, X-, Y-, or Z-direction displacement dependent.
  - Periodicity

    Specify this if the X-axis type is step ID dependent.
  - Phase

    Specify this if the X-axis type is rotation angle dependent.

3-16-18. Macro Component

- It is used when handling multiple circuit components as one component.
- The organization content of the macro component can freely be determined. Moreover, the macro component which can be utilized as well is loaded as the default.
- There is no need to set one component type as the macro component. Set, however, the circuit component type for the individual circuit component which organizes the macro component.

▶▶ Reference

- For more information about the cage component, lap winding (brush motor), and wave winding (brush motor) in JMAG-Studio, see Chapter 8 “Creating an External Circuit” in the JMAG-Studio User's Manual.
- For more information about the cage component, lap winding (brush motor), and wave winding (brush motor) in JMAG-Designer, see Chapter 35. “Circuit” in the JMAG-Designer User's Manual.

3-16-19. Electric Potential Probe Component

- This is used in the case of indicating voltages in places where no components exist, for instance, in-between connections. Arrange the electric potential probe at places where you wish to obtain the voltage in the circuit drawing.
- In the circuit connection drawing, it is possible to check the voltage at the place where the respective elements are arranged.

3-16-20. Current Probe Component

- This is used in case of indicating current value in places where no components exist, for instance, in-between connections. Arrange the current probe at places where you wish to obtain the voltage in the circuit drawing.
- In the circuit connection drawing, it is possible to check the current at the place where the respective components are arranged.

3-16-21. Ground Component

- It is used when wanting to ground the voltage.
- Specify either an electric potential source (one terminal) component or a ground component when using circuits.

3-16-22. Circuit Component Types and Settings

Type	Type number	Items
FEM coil	100	FEM coil group ID, Resistance value
FEM conductor	200	FEM conductor group ID
User subroutine	300	User variables (negative integer)
Resistor (constant)	-100	Resistance value
Resistor (time dependency)	-101	Relation point sequence for time and resistance
Resistor (rotation angle dependency)	-102	Relation point sequence for rotation angle and resistance
Resistor (X-axis displacement dependency)	-103	Relation point sequence for X-axis displacement and resistance
Resistor (Y-axis displacement dependency)	-104	Relation point sequence for Y-axis displacement and resistance
Resistor (Z-axis displacement dependency)	-105	Relation point sequence for Z-axis displacement and resistance
Resistor (voltage dependency)	-106	Point sequence of voltage dependent resistance, reference of electric probe component or reference terminal, initial resistance
Resistor (electric current dependency)	-107	Point sequence of current dependent resistance, reference component, initial resistance
Resistor (distance dependency)	-116	Point sequence of the distance-resistance relationship
Capacitor	-200	Capacitance value
Coil	-300	Inductance value
Switch (time dependency)	-401	Relation point sequence for time and open/close
Switch (rotation angle dependency)	-402	Relation point sequence for rotation angle and open/close
Switch (X-axis displacement dependency)	-403	Relation point sequence for X-axis displacement and open/close
Switch (Y-axis displacement dependency)	-404	Relation point sequence for Y-axis displacement and open/close
Switch (Z-axis displacement dependency)	-405	Relation point sequence for Z-axis displacement and open/close
Switch (voltage dependency)	-406	Point sequence of voltage dependent switching, reference of electric probe component or reference terminal, initial resistance

Type	Type number	Items
Switch (electric current dependency)	-407	Point sequence of current dependent switching, reference component, initial state
Switch (step dependency)	-408	Relation point sequence for step number and open/close
Switch (distance dependency)	-409	Table of distance-on/off relationship or parameters
Diode	-5	Voltage-current point sequence or coefficient and saturated current, upper and lower limits of impedance
Electric potential source (1 terminal)	-600	Voltage table of dependence upon time etc., or function definition by the parameter
Current source	-700	Current table of dependence upon time etc., or function definition by the parameter
Electric potential source (2 terminals)	-600	Voltage table of dependence upon time etc., or function definition by the parameter
Power source	-	Constant or time dependency for electrical power, maximum voltage, or maximum current can be specified.
Brush	-1200	Number of Brushes, number of commutator, slit width between commutators, initial position of brush, contact resistance of brush and commutator, arc voltage
3phase electric potential source	-1500	Phase difference among U-phase, V-phase, and W-phase, amplitude, frequency, U-Phase
3phase current source	-1600	Phase difference among U-phase, V-phase, and W-phase, amplitude, frequency, U-phase
Reference conductor	-1800	Referenced FEM conductor, periodicity
Reference coil	-1900	Referenced FEM coil, periodicity
Potential probe	-1300	None
Current probe	-1400	None
Grand	-	None

3-17. Motion (Rotation/Translation)

▶▶ Reference

- For more information, see "Motion" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see the "Translation Motion" and "Rotation Motion" sections of Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-17-1. Function

Performs an analysis for the case where a subject changes its position in each analyzing step. This condition enables analysis to be performed including the effects of an induced electromotive force.

In the JMAG, the relative positional relation of the motion region and other regions categorize motion into two types.

- The relative location of meshes changes:

  The relative location of meshes varies with motion.
  - This is appropriate for general motion.
  - This condition must be set when dealing with relative motion such as a motor or actuator.
  - Meshes actually move in this case.
- The relative location of meshes does not change:

  The moving region is constant against the motion direction. In other words, moving status can be represented without changing the relative location of meshes.
  - This can be employed only when the velocity and relative location of an object do not vary with time.

    For example) this is appropriate for analyzing an eddy current when a uniform and long conductive plate in the direction of movement is moving at a constant speed in a stationary magnetic field.

    Although meshes do not actually move in this case, consequently, eddy currents are generated since physical quantities are changed in consideration of the velocity.

■ Related items

- Step / frequency
- External circuit link
- Slide
- Generate mesh while running analysis
- User subroutine: *usrfma*
- User subroutine: *usrfma2*

▸▸ Reference

For more information about generating a mesh while running an analysis, see Chapter 5 “Generating Mesh” in the JMAG Studio User's Manual.

■ Targets

- Element group of moving part

■ Settings

- Types of motion
  - Rotational motion
    - Coordinate system of one point on the rotation axis.
    - Direction vector of rotating axis
    - Defining method of motion
      - Displacement specification (velocity, step displacement amount, cumulative displacement amount)
      - Equation of motion
  - Translational motion
    - Direction of motion
    - Defining method of motion
      - Displacement specification (velocity, step displacement amount, cumulative displacement amount)
      - Equation of motion
  - In case user subroutine usrfma is used
    - Identification number of user subroutine (negative integer)
    - Coordinate system of one point on the rotation axis.
    - Direction vector of rotation axis
    - Direction of motion

- Coupling with external circuit simulator
  * Only effective in Transient response magnetic field analysis (TR, DP)
- Eccentricity of rotation axis and motion region
- Flag that does not accompany displacement of mesh.

■ Available modules

- All modules except static analysis (ST)

■ Output items

- Displacement (m)

3-17-2. Important Notes Regarding the Use of This Function

- Motion involving transformation of magnetic and conductor cannot be handled.
- The rotation motion condition cannot be used for an axisymmetric analysis.
- Motion regions can be grouped. Multiple groups can be set.

3-17-2-1. Types of Motion

JMAG can analyze the following motions.

■ Rotational motion

A motion in which a body rotates around a certain axis.

■ Translational motion

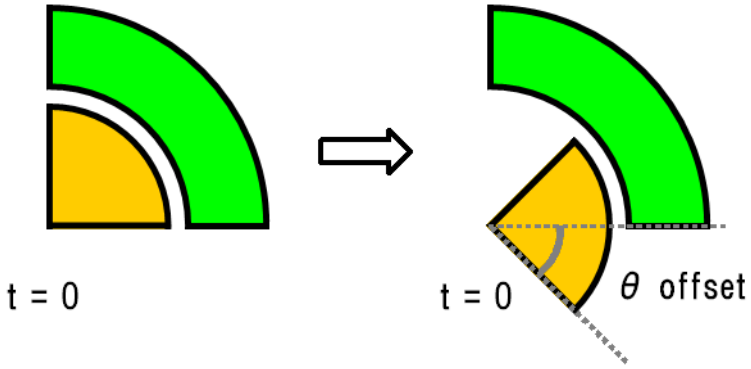
A motion in which a linear movement is made in a certain direction.

Important **This specification will apply whether the definition method is displacement specification, equation of motion, or user subroutine.**

3-17-2-2. Initial Position

Initial position can be set by specifying the offset with respect to the rotor (the mover)'s position in a created FEM model.

With this setting of initial position, the analysis starts with the rotor (mover) offset to the specified position. In a synchronous motor, for example, changing the initial position without changing the current phase is the same as changing the current phase overall.



Example: Initial position

- The displacement from the initial position is referenced in rotation angle dependence, displacement dependence of conditions and circuit components.
- The result model is shown in the initial position and the angles and distance in the graph are the displacement from the initial position.
- For rotation, specify the angle with the unit (deg). For translation, specify the distance in terms of three components (x, y, z) with the unit (m).
- When linking to an external circuit simulator, or using the magnetic field analysis solver as a DLL file, the initial position is not taken into account.
- If the offset specified with respect to the initial position causes the mover to contact the stator, the setting is invalid.

Important **Initial Position is a parameter different from Initial Displacement in the motion equation. Initial Displacement represents the initial status of elasticity, and therefore is not affected by changing the initial position in this setting. Spring constant is applied depending on the displacement of the mover.**

Important **Changing the initial position in this setting does not affect the initial position of a brush and commutator.**

3-17-2-3. Definition of Motion

The conditions of motion are defined using one of the methods below.

- Displacement
  - Method of specifying velocity (rotational velocity) at each time.
  - Method of specifying distance (rotational amount) of each step.
  - Method of specifying cumulative distance (rotational amount) at each time.
- Equation of motion
  - Based on the time intervals set by the step conditions, a method of solving the equation of motion (time specification).
  - A method of solving the equation of motion by setting displacement amount (constant) per 1 step with the step conditions (displacement specification).
- User subroutine
  - Method of specifying motion using the user subroutine *usrfma*.

Important **The transient response analysis (TR, DP) applies to all methods mentioned above.**

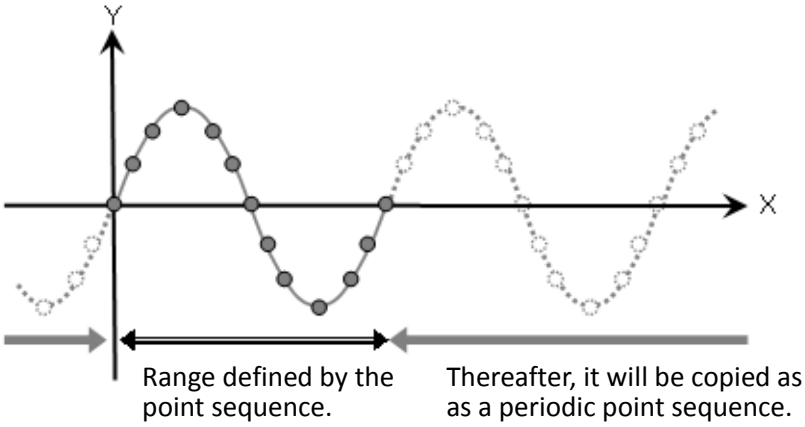
Important **The frequency response analysis (FQ) applies to only movement specification of constant displacement. Provided that in case it is coupled with thermal analysis (HT), Velocity Specification can also be selected.**

3-17-2-4. Displacement

■ Define the velocity with point sequence

- Enter two or more point sequences such as (T1, Y1), (T2, Y2). In this case, T1 and T2 must meet the conditions T1<Tmin<T2 and T1<Tmax<T2 regarding the minimum value Tmin and the maximum value Tmax of the X-axis parameters that are referred to in analysis.

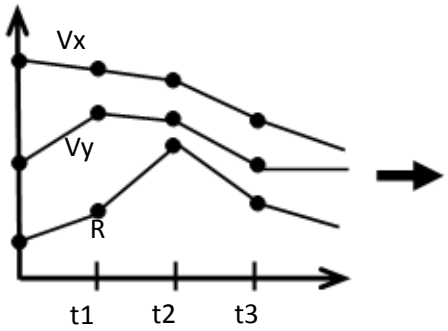
- When the [Periodic Point Sequence] check box is selected, in case the X-axis value exceeds the range defined by the point sequence data during the calculation, the point sequence will be copied periodically and be referred to.



If the [Periodic Point Sequence] check box is selected:

When the [Periodic Point Sequence] check box is cleared, in case an X-axis value that exceeds the range defined by the point sequence data is referred to during the calculation, a warning will be output to the solver report.

- In case of referring to an X-axis value that is outside of the range of a point sequence, the data will be interpolated automatically by deeming that the data is a periodic point sequence. In case it is not desirable to handle the data as periodic, by clearing the [Periodic Point Sequence] check box, it is possible to output a warning.
- The values set by the point sequence are approximated by a broken line within the program.
- The time specified by a point sequence doe not necessarily coincide with the analysis time specified in the step data.



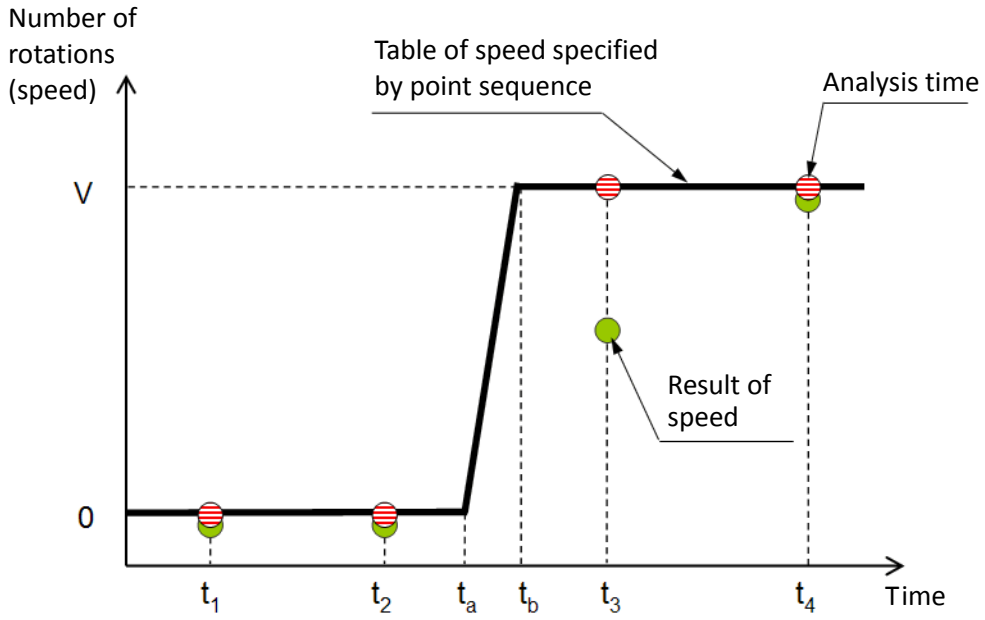
t	Vx	Vy	R
t1	Vx1	Vy1	R1
t2	Vx2	Vy2	R2
t3	Vx3	Vy3	R3
t4	-	-	-
-	-	-	-

Example of velocity point sequence

- The following is the value of speed exported to the result file when specifying speed using displacement (point sequence).

  i. The displacement between the steps for time is integrated.

  ii. The speed is determined by dividing the displacement in widths of time interval.



The speed V_{t23} between sections t_2 to t_3 is derived by using the displacement d_{t23} in the same direction.

$$d_{t23} = d_{t2a} + d_{tab} + d_{tb3}$$

$$V_{t23} = d_{t23} / (t_3 - t_2)$$

■ Define the distance with point sequence

- If displacement until the next step is specified, the unit of the X component of the point sequence is referred to by the step number. For the Y component of the point sequence, it should be specified for each step. Note that the displacement for the first step is zero. The object begins to move from the 2nd step.
- Even if displacement until the next step has been specified, enter the correct analysis time as the time of step condition.
- If all electric conductivities or velocities in a model are zero, the analysis becomes identical to a static analysis.

3-17-2-5. Equation of Motion

An equation of motion is expressed as follows.

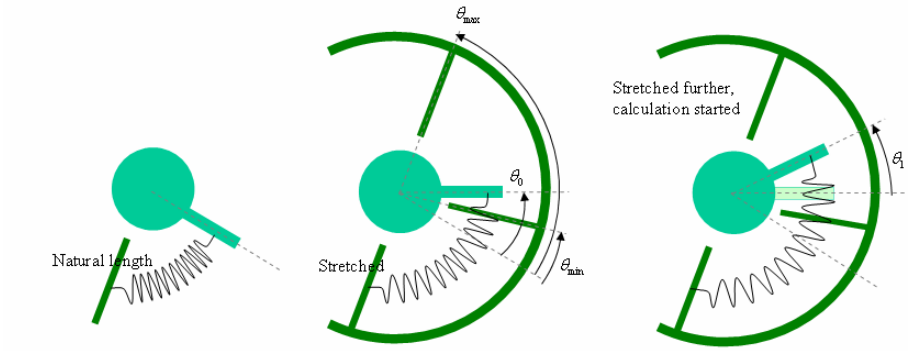
■ Rotational motion

$$T = J(\ddot{\theta}) + B(\dot{\theta}) + K(\theta - \theta_0 - \theta_1) + \tau$$

- T: Torque (N·m)
- J: Inertia moment (kg·m^2)
- B: Relaxation constant (N·m/(deg/sec))
- K: Spring constant (N·m/deg)
- $\ddot{\theta}$: Acceleration of rotation angle (deg/sec^2)
- $\dot{\theta}$: Velocity of rotation angle (deg/sec)
- θ : Extension of the spring (deg)
- θ_0 : Initial displacement, the extension of the spring at the initial model position (deg)
- θ_1 : Initial position (deg)
- τ : Friction torque (N·m)

- The extension of the spring is determined according to the initial displacement and initial position, which are separate parameters. For the initial displacement, specify the extension of the spring when the moving part is initially created.

  For example, in the case of the model in the middle of the figure shown below, specify the stretched length θ_0 as the initial displacement because the spring is stretched from its natural length as shown on the left side of the figure. If you want to start the calculation from the position shown on the right side of the figure below, enter θ_1 for the initial position. If a value other than zero is specified for the initial position, the analysis will start with the moving part displaced by the specified angle.



Concept of spring in calculation

Created FEM model
Initial displacement: θ_0
Initial position: θ_1
Range of motion: θ_{min} - θ_{max}

Initial position shifted to the first step

- The range of motion is defined with respect to the position where the spring has its natural length. The motion is controlled within the specified range of motion. If the rotor reaches the upper or lower limit of the range, the motion will stop at the position unless a force is applied in the opposite direction.
- Note the following when specifying the inertia moment, spring constant, and attenuation constant.
  - For the full model, enter the value for the full model.
  - If you turn on the full model conversion function for a partial model, enter the value for the full model.
  - If you turn off the full model conversion function for a partial model, enter the value obtained by dividing the full model value by the number of divisions.

- If you set the inertia moment to [Automatic], the inertia moment will be calculated based on the density specified for the material and the model conversion setting to be used for the calculation.

Reference
For details of the full model conversion, see P.283 "3-40. Full Model Conversion".

- You can calculate the time by specifying the displacement (deg) of the rotor at one step in the step control. The time required for the rotor to move within the specified range from one end to the other will be calculated during the analysis. The calculation stops if the rotor cannot move to the extent of the specified range or the time needed for the rotation exceeds the maximum time allowed. When the rotor can move either positive or negative direction, the time it takes to reach whichever first will be calculated.

Reference
For details of the step control, see P.272 "3-38. Step Control".

$$T = kT1 + T2$$

  - T1: Torque obtained by the referenced electromagnetic force and torque calculation condition (N·m)
  - k: Correction multiplier for torque T1
  - T2: Load torque (N·m)
- The positive direction of load torque is counterclockwise about the rotation axis of the motion condition. To set it as a load torque, select the sign opposite of the rotation direction. For example, when the rotor moves counterclockwise about the rotation axis, in the positive direction, specify the negative value to apply a load.
- Load torque can be specified by the settings of constant, time-dependent point sequence, rotation angle (mechanical angle) dependent point sequence and angular velocity dependent point sequence.

- To define the time-dependent load torque, specify a value including the initial displacement (θ_0).

  Displacement (angle) = $\theta - \theta_0 - \theta_1$

  θ : Extension of the spring (deg)

  θ_0 : Initial displacement (the extension of the spring at the initial model position (deg))

  θ_1 : Initial position (deg)

- You can specify load torque at each time (step) using the user subroutine *usrfm2*.

  $J = pJ1 + J2$

  - J1: Inertia moment of target region ($kg \cdot m^2$)
  - p, J2: Correction multiplier and correction value for J1

  $\tau = \mu \cdot r \cdot Fc$

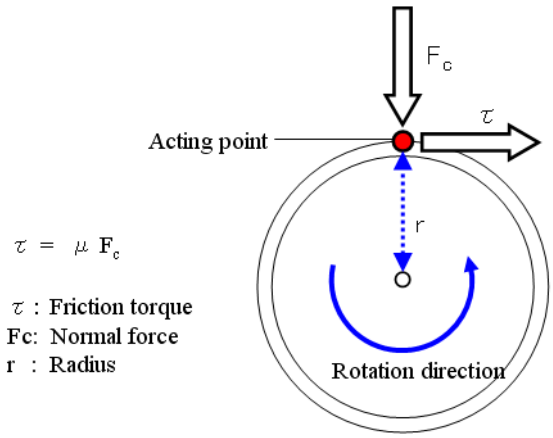
  - μ: Friction coefficient (static, dynamic)
  - Fc: Normal force (N)
  - R: Radius from the normal force acting point (m)

- A friction coefficient is expressed as follows:

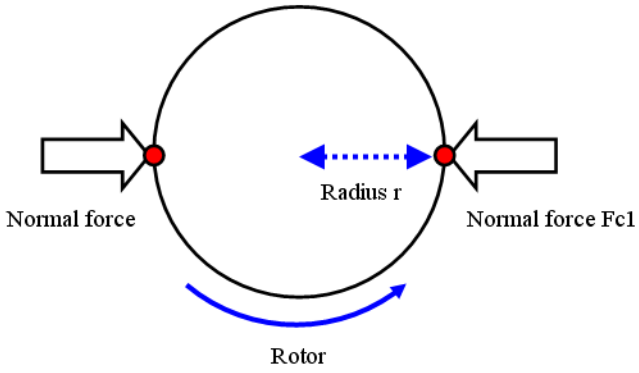
  $\mu = \mu_d + (\mu_s - \mu_d)e^{-c|v|}$

  where v is the velocity (m/sec) obtained by multiplying the angle speed (rad/sec) with the radius of the friction surface R (m). The exponent c represents the relaxation constant (sec/m) to ensure a smooth connection between static and dynamic frictions. This is fixed to 1.0.

- The friction torque is determined by the normal force acting on the friction surface and the radius from the point where the normal force is applied.



One acting point: (τ: friction torque, Fc: normal force, r: radius)



As shown in the figure above, when the normal force is applied to two points, set it as the sum of normal force Fc1 + Fc2.

Both Fc1 and Fc2 in the above figure are positive values.

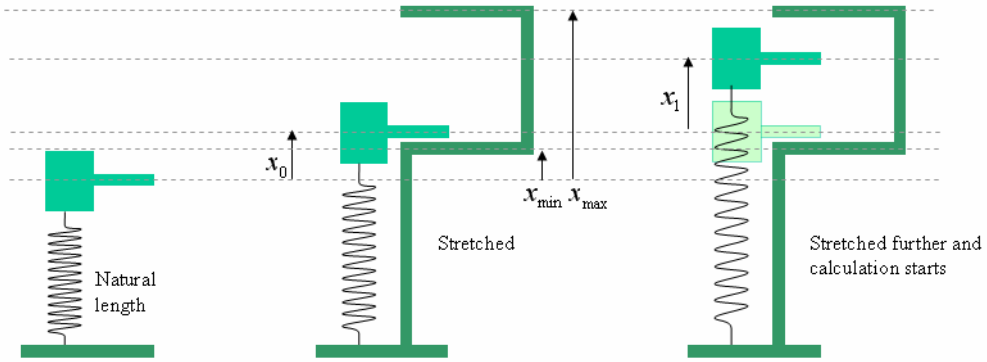
■ Translation motion

$$F = M(\ddot{x}) + B(\dot{x}) + K(x - x_0 - x_1) + f$$

- F: External force (N)
- M: Mass (kg)
- B: Relaxation constant (N/(m/sec))
- K: Spring constant (N/m)
- $\ddot{x}$: Acceleration in the translation direction (m/sec^2)
- $\dot{x}$: Velocity in the translation direction (m/sec)
- x : Extension of the spring (m)
- x_0 : Initial displacement, the extension of the spring at the original position of the moving part (m)
- x_1 : Initial position (m)
- f: Friction force (N)

- The extension of the spring is determined according to the initial displacement and initial position, which are separate parameters. For the initial displacement, specify the extension of the spring when the moving part is initially created.

  For example, in the case of the model in the middle of the figure shown below, specify the stretched length X_0 as the initial displacement because the spring is stretched from its natural length as shown on the left side of the figure. If you want to start the calculation from the position shown on the right side of the figure below, enter X_1 for the initial position. If a value other than zero is specified for the initial position, the analysis will start with the moving part displaced by the specified distance.



Concept of spring in calculation

Created FEM model
Initial displacement: X_0
Initial position: X_1
Range of motion: X_{min} - X_{max}

Initial position shifted to the first step

- The range of rotation is determined with respect to the initial position of the mover where the spring connected to it has no tension.

  If the rotor reaches the upper or lower limit of the range, the motion will stop at the position unless a force is applied in the opposite direction.

  In the case of a model with the spring stretched by 0.1 m, and the moving part that moves only 1 m in the positive direction from the position where the extension of the spring is zero.

  Initial displacement: 0.1m

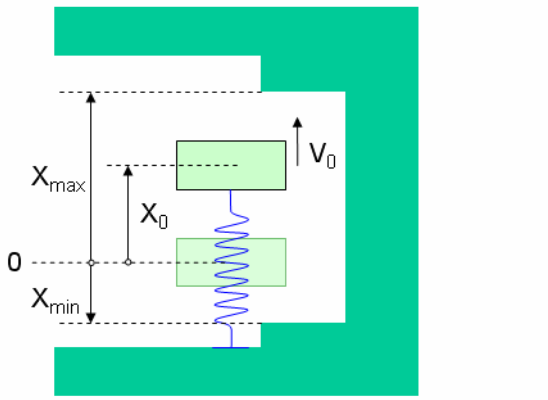
  Rotation range in the positive direction: Xmax: 1.0m

  Rotation range in the negative direction: Xmin: 0.0m

In this case, the mover has the rotation range of:

0.0 <= X <= 1.0 with respect to the spring, or

The range of the motion is - 0.1 <= X <= 0.9 with respect to the original model position.



Initial position X_1: The position of the mover at the first step

Initial displacement X_0: The state of the spring at the initial position

(x_0 = 0 natural length) (X_0 < 0 compressed) (X_0 > 0 stretched)

- Note the following when specifying the mass, spring constant, and attenuation constant.
  - For the full model, enter the value for the full model.
  - If you turn on the full model conversion function for a partial model, enter the value for the full model.
  - If you turn off the full model conversion function for a partial model, enter the value obtained by dividing the full model value by the number of divisions.
  - If you set the mass to be automatically calculated, it will be calculated based on the settings of the material density and the model conversion, and used for the analysis.

Reference

For details of the full model conversion, see P.283 "3-40. Full Model Conversion".

- You can calculate the time by specifying the displacement (m) of the mover at one step in the step control. The time required for the mover to move within the specified range from one end to the other will be calculated during the analysis. The calculation stops if the mover cannot move to the extent of the specified range or the time needed for the rotation exceeds the maximum time allowed. When the mover can move either positive or negative direction, the time it takes to reach whichever first will be calculated.

  Reference

  For details of the step control, see P.272 "3-38. Step Control".

$$F = kF1 + F2$$

  - F1: Electromagnetic force (N) obtained by the referenced electromagnetic force and torque calculation condition
  - k: Correction multiplier for electromagnetic force F1
  - F2: Additional force (N) other than electromagnetic force F1 obtained by the condition
- The positive direction of additional force F2 is the motion direction specified for the motion condition. To set it as a load force, select the sign opposite of the motion direction.
- Additional force F2 can be specified by the settings of constant, time-dependent point sequence, position dependency point sequence and velocity dependent point sequence.
- To define the displacement-dependent additional force F2, specify a value including the initial displacement (x_0) for the displacement (distance).

  Displacement (distance) = $x - x_0 - x_1$

  x : Extension of the spring (m)

  x_0 : Initial displacement (the extension of the spring at the initial model position (m))

  x_1 : Initial position (m)

- You can specify additional force F2 at each time (step) using the user subroutine *usrfm2*.

  $M = pM1 + M2$

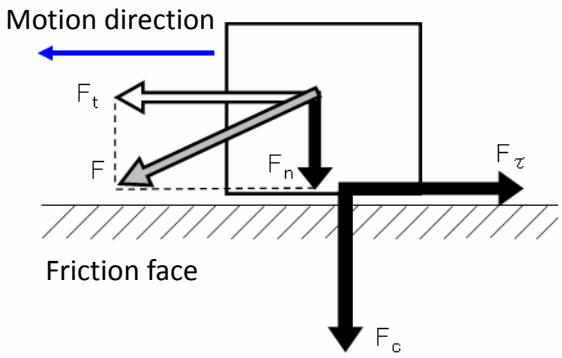
  - M1: Mass in target region (kg)
  - p, M2: Correction multiplier and correction value for M1

  $f = \mu(Fc + \chi Fn)$

  - μ: Friction coefficient (static, dynamic)
  - Fc: Normal force (N)
  - Fn, χ : Normal component with respect to the friction surface and correction multiplier of electromagnetic force obtained by the referenced electromagnetic force and torque calculation condition
- A friction coefficient is expressed as follows.

  $$\mu = \mu_d + (\mu_s - \mu_d)e^{-c|v|}$$

  where v is the velocity (m/sec) and the exponent c represents the relaxation constant (sec/m) to ensure a smooth transition between static and dynamic frictions. This is fixed to 1.0.
- The friction force is determined by the normal force and normal component of the electromagnetic force on the friction surface.



$F_\tau = \mu (F_c + F_n)$

F: electromagnetic force
F_c: normal force
F_n: normal component of electromagnetic force F
F_τ: parallel component of electromagnetic force F

- The normal component of the electromagnetic force is determined by the referenced electromagnetic force and the normal direction perpendicular to the friction surface specified by the vector or the local coordinate system.

3-17-2-6. User Subroutine

- When describing a motional state by using a subroutine, use the subroutine *usrfma*.

3-17-2-7. Linking to an External Circuit Simulator

When direct coupling is conducted with an external circuit simulator designated by JMAG, this is used when conditions of motion are made at the input or output terminal at the external circuit simulator.

The following are the procedures.

■ JMAG-Studio

Select the [Link to External Circuit Simulator] check box in the [Motion] condition to set the external link conditions. The following procedure is used to set the external circuit link condition.

- The displacement is used as the input terminal and the output terminal when [Displacement] is selected in the [Motion] condition.
- The displacement is used as the output terminal when [Equation of Motion] is selected in the [Motion] condition.
- The values for the X, Y, and Z components can be used when [Translation] is selected.
- The values for the theta component can be used when [Rotation] is selected.

■ JMAG-Designer

Set one or two external circuit link conditions after setting the [Motion] condition.

- Information for the displacement can be used in an external circuit simulator when [External Circuit Link 1] is used.
  - The displacement is used as the input terminal (IN) and the output terminal (OUT) when [Displacement (Link to External Simulator)] is selected in the [Motion] condition.
  - The displacement is used as the output terminal (OUT) when [Equation of Motion] is selected in the [Motion] condition.
  - The values for the X, Y, and Z components can be used when [Translation] is selected.
  - The values for the theta component can be used when [Rotation] is selected.

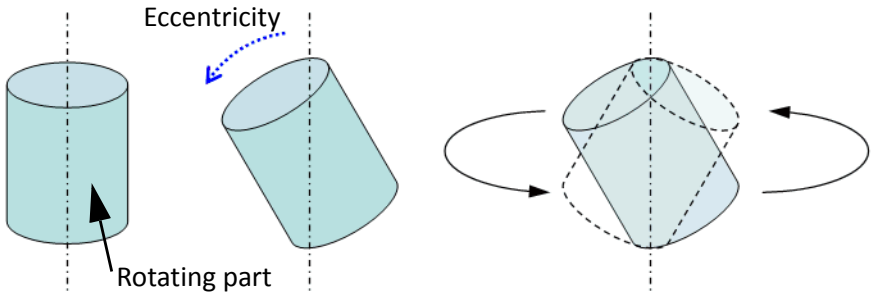
- The motion condition is set to define the information for the mover when using [External Circuit Link 2]

Important **The external circuit simulator link can only be used for the transient response analysis.**

Important **The external circuit link condition needs to be specified**

3-17-2-8. Setting Eccentricity

The eccentricity for a model can be used.



Example: Eccentricity is set to a rotating part

- This function can be used for 2D and 3D models.
- This function can be used for a model with rotation motion.
- Multiple motion conditions can be used. However, this function can only be used for models with fixed regions that are in between different motion regions (models with motion regions that are not continuous).
- The periodic boundary condition cannot be used due to loss of rotation periodicity when using the eccentricity function.
- The eccentricity for the rotating part and rotation axis need to be set when running the motion region and rotation axis together. The rotation axis is not modified by setting only the eccentricity of a rotating part.
- Specify the eccentricity amount so that the gap width G2 after eccentricity is more than 20% of the gap width G1 before eccentricity.
  Gap width G2 after eccentricity for a 2D model that is not tilted is calculated as below using the offset for a rotating part (Dmx, Dmy, 0) and the offset for a rotation axis (Dax, Day, 0).

$$G_2 = G_1 - \sqrt{(D_{mx} - D_{ax})^2 + (D_{my} - D_{ay})^2} - \sqrt{D_{ax}^2 + D_{ay}^2}$$

■ Mesh generation during usage of the eccentricity function (JMAG-Designer)

- Generates mesh using eccentricity during an analysis (mesh cannot be created when the eccentricity function is set before running an analysis).
- The eccentricity function can be used regardless of the mesh type (standard mesh, slide mesh, remesh at each step). However, the following should be noted.
  - An air region such as a gap area cannot be modeled when a mesh is created manually before running an analysis and the slide mesh or the remesh at each step function is selected.
  - Specify at least 4 as the value in the number of radial divisions when a slide mesh is selected as the mesh type.
- The values for each parameter in the rotation axis for a motion condition or the magnetization vector of a magnet are updated in response to the eccentricity amount after generating a mesh.

■ Mesh generation during usage of the eccentricity function (JMAG-Studio)

- The timing when the eccentricity function is applied to an analysis model occurs in the following three ways.
  - Generating mesh by using the function to create a standard mesh and air region before running an analysis
  - Creating a slide mesh (rotational) before running an analysis
  - A mesh is generated automatically during an analysis
- The values for each parameter in the rotation axis for a motion condition or the magnetization vector of a magnet are updated in response to the eccentricity amount after generating a mesh.

■ Generating mesh by using the standard meshing and air region meshing or generating mesh by using slide mesh (rotational)

- The mesh is generated using eccentricity when the [Use Eccentricity] check box is selected in the motion condition (rotational) during mesh generation.
- Specify at least 4 as the value in the number of radial divisions when generating a slide mesh using eccentricity.
- The [Use Eccentricity] check box is cleared if a mesh is generated using the function to automatically generate a standard mesh with air region or a slide mesh is generated. An analysis is run using eccentricity when the above settings are specified.
- The mesh may be broken if eccentricity is applied on a model during an analysis when the eccentricity function is set for an analysis model with air region by using the function to automatically generate a standard mesh and air region.

■ Generating a mesh automatically during analysis

- An air region cannot be created before an analysis when generating a mesh during an analysis and eccentricity is used on a model.
  An analysis can be run without creating an air region by using one of the following functions.
  - Select [FEM+BEM] as the calculation method (page 208)
  - Using the function to remesh at each step (patch mesh function)
  - Using the function to generate slide mesh with the function to remesh at each step

▶▶ Reference

For more information about functions related to mesh, see the following explanation.

- See "Chapter 5 Generating Mesh," in the JMAG-Studio User's Manual for JMAG-Studio.
- See "Chapter 15 Generating Mesh on Geometry Editor (Manual)," and "Chapter 36 Generating Mesh (Automatic)" in the JMAG-Designer User's Manual.

■ Setting eccentricity to a rotating part

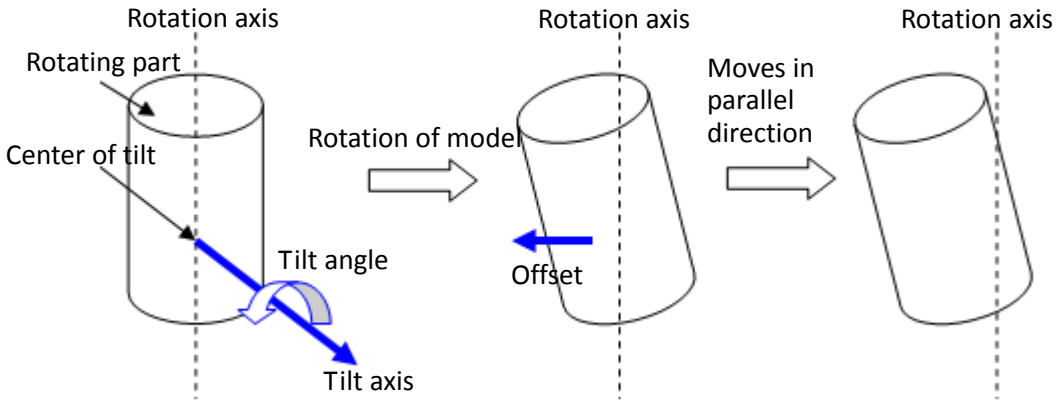
Specify the following settings.

- Center of tilt axis of rotating part (Omx, Omy, Omz)
- Direction of tilt axis of rotating part (Vmx, Vmy, Vmz)
- Tilt angle (Wm)
- Offset (Dmx, Dmy, Dmz)

MEMO The center axis that a rotating part revolves around is the tilt axis. This is not a rotation axis after eccentricity.

JMAG creates eccentricity on a rotating part with the following.

- Rotate the rotating part Wm around the spinning axis in which the direction is (Vmx, Vmy, Vmz) and passes through (Omx, Omy, Omz). The right screw is spinning in the Wm direction and moving towards the (Vmx, Vmy, Vmz) direction.
- The rotating part is moved (Vmx, Vmy, Vmz) in a parallel direction after rotation.



Eccentricity (Rotating Part)

■ Setting the rotation axis to eccentricity

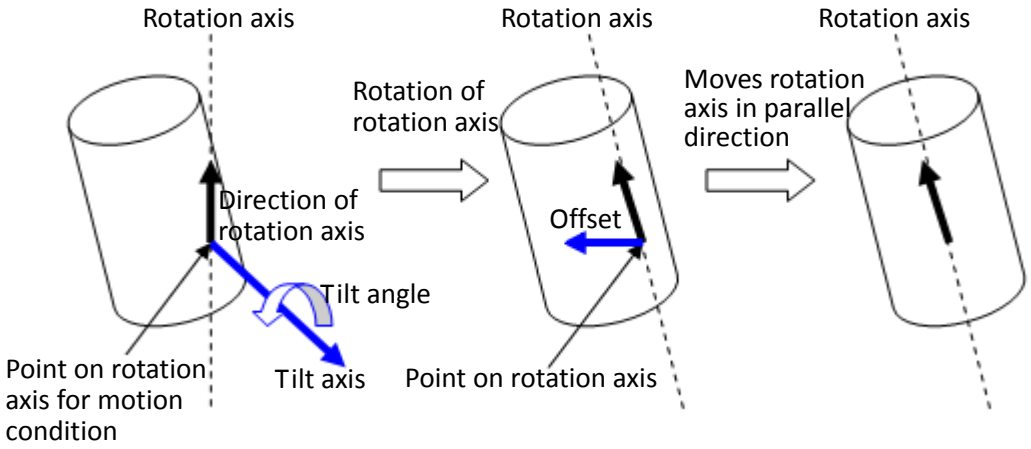
Specify the following settings.

- Direction of tilt axis (Vax, Vay, Vaz)
- Tilt angle (Wa)
- Offset (Dax, Day, Daz)

MEMO The center axis that rotates the rotation axis of a motion condition is the tilt axis. This is not a rotation axis after eccentricity.

JMAG moves the rotation axis of a motion condition with the following.

- Revolves Wa around Vax, Vay and Vaz in the direction of the rotation axis.The right screw is spinning in the Wa direction and moving towards the (Vax, Vay, Vaz) direction.
- A point on the rotation axis moves in (Dax, Day, Daz).



Eccentricity (Rotation axis of motion condition)

3-17-2-9. Setting virtual movement (JMAG-Studio)

In case an analysis in which the relative position of the mesh does not change is conducted, select the [Virtual Movement] check box in the [Motion] dialog box. On the contrary, in case an analysis in which the relative position of the mesh changes, clear this check box.

3-17-3. Generating Mesh with the Motion Condition Set

When using motion condition, it is required to modify the mesh generation method according to the model dimension and motion type.

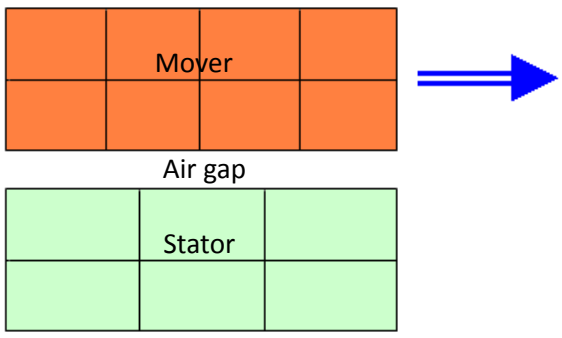
Run the most appropriate one from among the following methods.

- Applying FEM+BEM solver
- Using the slide condition
- Use the patch mesh feature (generate mesh while running analysis)
- Considering displacement without moving mesh

3-17-3-1. Using FEM+BEM Solver

By using the finite element method (FEM) in combination with the boundary element method (BEM) in the 2D model, analysis can be run while moving the movable part of the mesh in accordance with the motion condition setting.

- This method corresponds to both rotational and translational motion.
- The equation of motion can be used.
- Although this method can be used for 2D analysis, it cannot be used for 3D analysis.
- Under this condition, creating elements in the air gap between the movable part and the fixed part is not required.
- The number of subdivided elements for the movable part and the fixed part need not be the same.



Subdivided element of the 2D model

- [Add a layer of air in the computational domain force and torque] can be set in mesh properties when this method is selected, and the force condition and torque condition is set.

▶▶ Reference

- For more information about the calculation method, see "3-37. Calculation Method" on page 270.

- For more information about the force condition and torque condition, see "3-19. Electromagnetic Force and Torque Calculation" on page 216.
- For more information about [Add a layer of air in the computational domain force and torque], see "9-2. Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (JMAG-Designer)" on page 767.

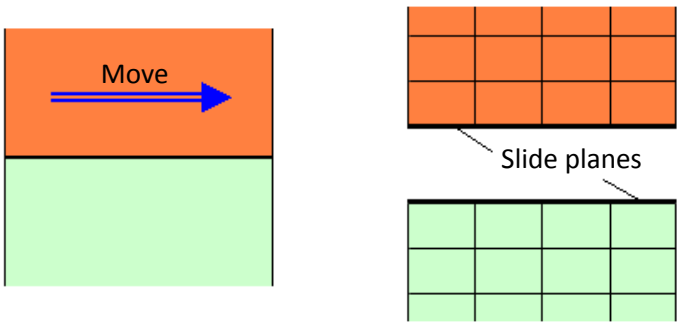
3-17-3-2. Using the Slide Condition

It is possible to create a slide plane between the movable and fixed parts, and vary the positions of the movable and fixed parts at every analysis step.

- This condition can be used for 2D and 3D analysis.
- The eddy currents are taken into account during analysis when this method is selected.
- The slide condition can be set by generating mesh manually.
- The slide condition is set automatically when the mesh, including the air region is generated, when the slide mesh option is used.

  ▶▶ Reference
  - For more information, see Chapter 5 “Creating Mesh” in the JMAG-Studio User's Manual.
  - For more information, see Chapter 36 “Generating Mesh (Automatic)” in the JMAG-Designer User's Manual.
- Generate the mesh taking care of the following points.
  - Divide elements on the slide plane in equal intervals with respect to the moving direction.
  - Make the number of elements on the slide plane of both the movable part and fixed part be equal.

- For all the analysis steps, each one of the elements on the first slide plane should correspond to an element on the second slide plane.

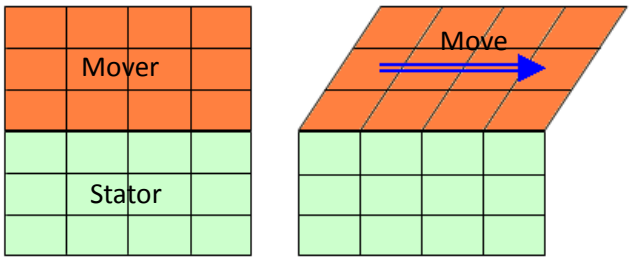


FEM model and slide plane

- Set each condition as follows.
  - Motion condition
    - The equation of motion can be used.
    - It does not matter even if displacement per step is larger than element size.
    - Set the displacement per step of an object as large as the element size on the slide plane.
  - Periodic boundary condition
    - This is required when setting the slide condition for the partial model.
  - Slide condition

    When setting the slide condition manually, be sure that nodes are not shared between the movable and fixed parts.

    Example: When an object sharing nodes between movable and fixed parts is moved without setting the slide plane.



If nodes are shared, changes occur in the mesh as it is dragged by the motion of the movable part.

3-17-3-3. Using the Patch Mesh Option

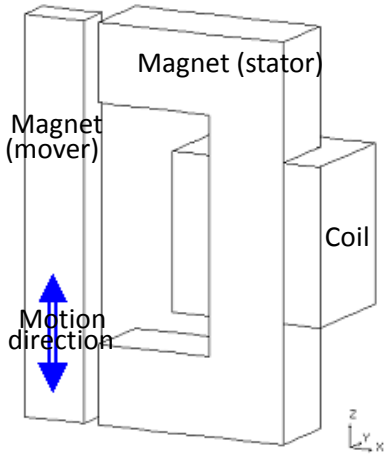
This is a method to automatically generate a mesh corresponding to the position of the movable part during analysis. With this method, elements are created automatically even for the air gap.

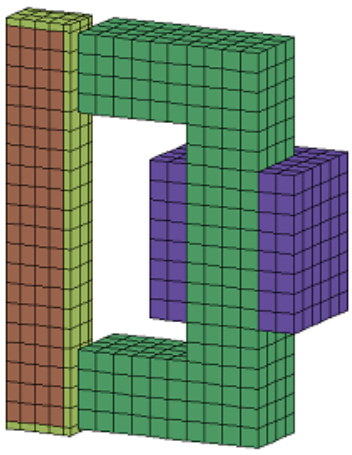
- In JMAG-Studio, select the [Generate Mesh While Running Analysis] check box in the [Automatic Generation] dialog box, and set a motion condition to use the patch mesh.
- In JMAG-Designer, select the [Remesh Model at Every Step] radio button in the [Mesh Properties] dialog box, and set a motion condition to use the patch mesh.
- The patch mesh function cannot be used in a 2D analysis. Select the direct method (FEM+BEM) as the calculation method instead.
- This option cannot be used for the analysis where the mover is in contact with the stator while it moves.

▶▶ Reference

- For more information, see Chapter 5 "Creating Mesh" in the JMAG-Studio User's Manual.
- For more information, see Chapter 36 "Generating Mesh (Automatic)" in the JMAG-Designer User's Manual.

■ Translational motion





Analysis model (model outline) Analysis model (mesh line)

* The analysis model is a 1/4-size model that was cut out on the YZ and ZX planes.

- Create an analysis model comprising the magnetic material and the conductor parts. Non-magnetic materials such as plastics have almost no effect on an analysis.

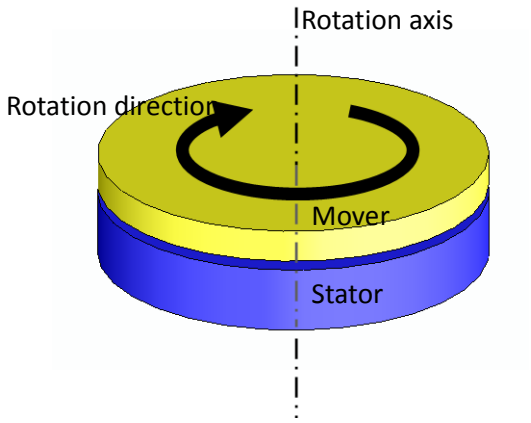
- Set each condition as follows.
  - Magnetomotive force
    Define the current or magnet with each condition or material property.
  - Boundary conditions (periodic boundary and symmetry boundary) and air layer cross section
    Conduct settings in the case of partial models.

Important **When air layers are created around a moving object as in the example, motion conditions must be set to include all the moving parts as well as the surrounding air layers. Slide conditions are not required to be set.**

■ Plane rotational motion

Rotational motion on plane gap surface as shown below is defined as “Plane rotation motion”.

Analysis can be performed using Finite Element Method without worrying about the position of rotating parts relative to static parts and consistency of the mesh on rotating and static parts by embedding tetrahedral elements into the spatial gap between movable part and static parts.



Plane rotational motion

MEMO Starting from version 8.1, it became possible to conduct patch mesh (arbitrary motion patch mesh execution) to partial models by setting the periodic boundary conditions.

- Set as follows when creating the mesh.
  - For JMAG-Studio, select the [Automatic Generation] radio button and the [Generate Mesh While Running Analysis] check box in the [Automatic Generation] dialog box.
  - For JMAG-Designer, select [Remesh Model at Each Step] in the [Mesh Properties] dialog box.

- This function can be utilized for full models or partial models. In the case of partial models, the following should be noted:
  - This does not correspond to axial reverse periodic boundary conditions and translational periodic boundary conditions.
- When conducting magnetization in JMAG-Studio, set [Magnetize When Running Analysis] to conduct magnetization during the analysis.
- Set each condition as follows.
  - Magnetomotive force
    Define current or magnet with each condition and material property.
  - Symmetry boundary condition
    Make settings in case of partial model.
  - Periodic boundary condition
    Make settings in case of partial models having rotational periodicity.

3-17-3-4. Displacement without Moving the Mesh

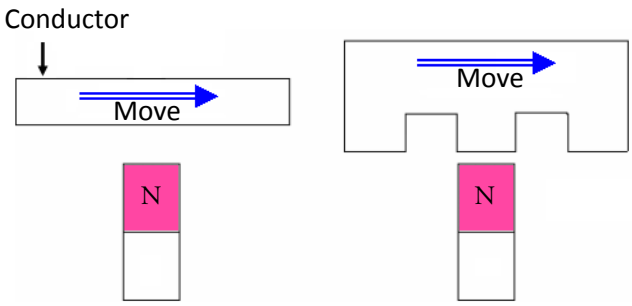
This method can be used when the movable part is constant with respect to the motion direction, i.e. the relative positions of the movable part and the fixed part do not change.

Considers the velocity electromotive force by displacing only the physical quantity, without making the mesh move.

- If this method is selected, the eddy current can be considered during the analysis.
- This method can be used for both 2D and 3D analyses.
  - In the TR (3D transient response analysis) module, a transient calculation will be continued until it reaches a steady state.
  - In the DP (2D and axial symmetric transient response analysis) module, a calculation can reach a steady state in a single step.

- This method can only be used when the velocity of the object and the relative positions do not vary with time.

  For example, this is appropriate for analyzing eddy currents when a uniform long conducting plate is moving at constant speed in a stationary magnetic field.



The relative location does not change The relative location changes

  - Set each condition as follows.
    - Motion condition
      - Select the [Virtual Movement] check box.
      - Set the motion velocity by the velocity point sequence.
    - Step condition
      - Set the time step of an analysis so that the displacement per step in the moving region is less than or equal to half of the minimum element in a moving direction. If the displacement per step is too large, the accuracy will be reduced.
      - In the DP module, set the time at 0.0 sec in the analysis.
  - When selecting this method is selected, generate the mesh with the following points in mind.
    - For motion of a rotational body, attach one or more air spaces on its circumference.
    - In the motion region, include the layer of air on the perimeter of the movable part.

3-18. Slide

▶▶ Reference

- For more information, see “Slide” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Slide” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-18-1. Function

This is used when the object to be analyzed changes in its location with each analysis step. This function enables displacements that are larger than the mesh size to be handled.

MEMO The slide condition is automatically set when the mesh is generated using the slide mesh option.

■ Related items

- Periodic boundary
- Motion * Mandatory
- Slide mesh

▶▶ Reference

For more information about the slide mesh option, see the following:

- Refer to the JMAG-Studio User's Manual Chapter 5 “Generating Mesh” for JMAG-Studio.
- Refer to the JMAG-Designer User's Manual Chapter 36 “Generating Mesh (Automatic)” for JMAG-Designer.

■ Targets

- First slide plane
- Second slide plane

■ Available modules

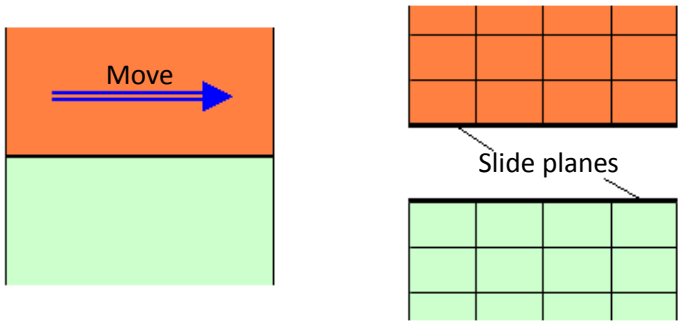
- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

■ Output items

- N/A

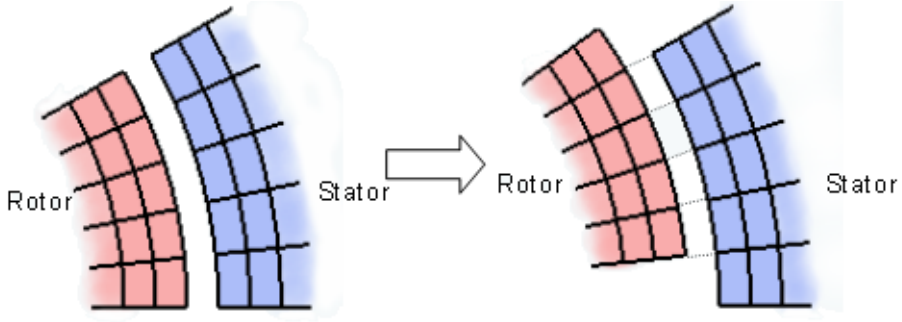
3-18-2. Important Notes Regarding the Use of This Function

- FEM models (not continuous) are created for two objects that are moving relatively. Boundaries of these objects are called the first slide plane and second slide plane, respectively. Either boundary can be the first plane.

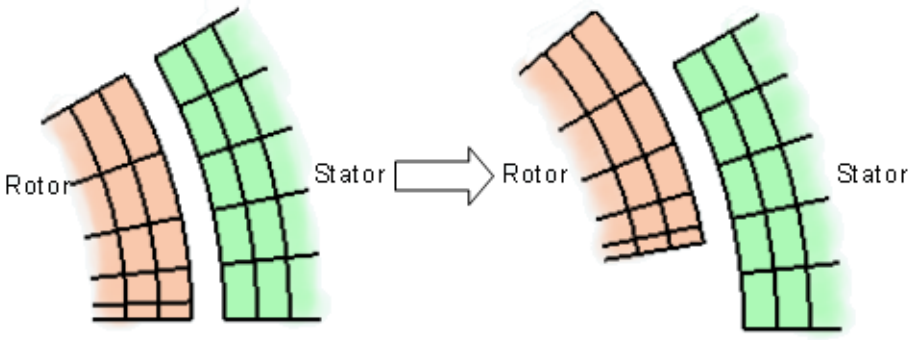


FEM model and slide plane

- It is necessary that the subdivided element of the slide surface is conducted at equal intervals against the movement direction.



Example 1. When subdivided element is made in equal intervals



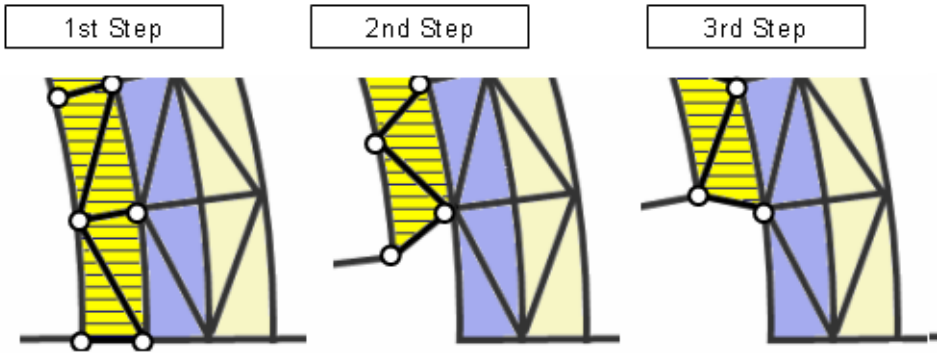
Example 2. When element division is not conducted at equal intervals

If the rotor side moves the equivalent of one element, the conformity of the division position is broken, and the analysis results will be incorrect.

- The number of elements on the first slide plane must be the same as that of the second, and also all elements of one slide must correspond to those on the other at each analysis step.

  MEMO To confirm elements, select in the following way: [Selection] > [Check Number of Selected Items] under the condition that a slide plane has been selected.

- In 2D analysis, this is available for rotational movement only.
- A slide plane cannot be set for the surface of a solid tetrahedral element.
- When the sliding conditions between the two conductors that are in contact are set, calculations in which one side of the conductor moves cannot be made. Make a thin air layer at the contact portion, and set the sliding conditions between the air layer and the contact surface of one of the conductors.
- If the subdivided element on the slide surface comes to a position that does not conform, only the nodes on the slide surface undergoes a forced displacement to a conforming position. By doing so, it becomes possible to make calculations even when the model comes to a position that does not conform such as when an equation of motion is used.



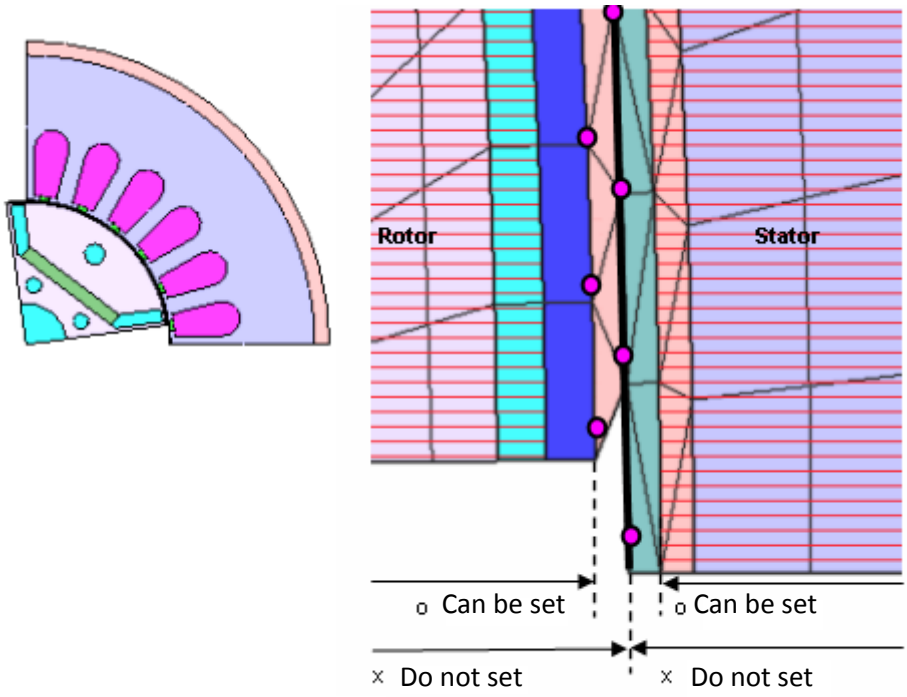
Element of the gap portion (In case only the nodes undergo forced displacement to conforming positions)

The nodes contained in the element having horizontal line patterns are moved to match the positions of the nodes in the adjoining element.

When running analysis of such non-conforming positions, pay attention to the following points.

- As for the slide surface, set it to element face of "air", solid plane, or element edge.
- Since the nodes on the slide surface undergo forced displacement so that they come to positions that conform, the elements will deform.
- In case the slide conditions are set to elements other than air, the calculation accuracy cannot be retained.

- Make it so that the calculation region of the electromagnetic force does not contain element that has slide surface set to it.



* The solid line portion is the slide surface

MEMO (In case of the following drawing, in order that the node ● is not included, do not include elements other than those of the horizontal line pattern in the setting object of "Electromagnetic Force and Torque Calculation".

- Make the subdivided element along the movement direction of the slide surface equal intervals.
- Prepare one set of slide surfaces for each gap.

3-19. Electromagnetic Force and Torque Calculation

▶▶ Reference

- For more information, see "Electromagnetic Force and Torque Calculation" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Force" and "Torque" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-19-1. Function

The electromagnetic force and torque calculation condition is used to calculate the force acting on magnetic materials and conductors in a magnetic field, and then calculate the torque about an axis using the force that is obtained.

■ Related items

- Material properties
- Motion
- External circuit link
- Mesh: Adds an air layer to the electromagnetic force and torque calculation area

■ Settings

- Calculation area for electromagnetic force and torque
- Point on the rotation axis
- Direction of the rotation axis
- Calculation method for electromagnetic force
- Link to an external circuit simulator *For transient response analysis (TR, DP) only

■ Available modules

- All of the modules of magnetic field analysis

■ Output items

- Nodal force (N)
- Surface force density (N/m^2)

- Electromagnetic force (N)
- Torque (N·m)
- Lorentz force (N)

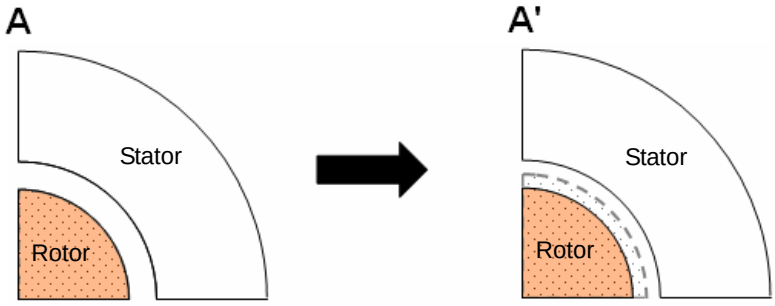
3-19-2. Important Notes Regarding the Use of This Function

- Multiple electromagnetic force and torque calculation conditions can be set for one model. The calculation area specified for each condition can be overlapped.
- The settings vary with the calculation method.
  - 3D analysis:
    This condition needs to be set for parts or elements when the nodal force method or Lorentz force method is selected, and for solid faces and element faces when the surface force method is selected.
  - 2D analysis:
    This condition needs to be set for parts or elements regardless of a calculation method that is selected.
- The torque around the rotation axis is calculated. For example, set [Point on Axis] to (0, 0, 0) and [Axis Direction] to (0, 0, 1) in the torque condition to calculate the torque for the Z-axis. The torque that is calculated is positive counterclockwise from the positive direction of the rotation axis.
- The electromagnetic force only in the modeled areas can be calculated. When the conversion to full model values is set, the calculation results are multiplied by the conversion factor specified for the periodic boundary and for other than periodic boundary. For a 2D analysis, the thickness of a model is also taken into account.
- The Lorentz force acting on a current is also calculated. The total amount of Lorentz force is automatically calculated for parts having materials where an electric conductivity is specified.
- In an axisymmetric analysis, torque is not calculated.
- In an axisymmetric analysis, only the X- and Y-component of electromagnetic force are obtained. The electromagnetic force in the radial direction is output as the X-component, and the electromagnetic force in the axial direction is output as the Y-component.
  In the evaluation of an entire model, the radial force is offset to zero, but it should be considered that the electromagnetic force is evaluated for the force expanding or contracting in the vicinity of the unit angle in one revolution.

- In a 2D analysis, the nodal force acting on the nodes in the calculation area and the surface force density acting on the outer circumference of the calculation area are obtained. The surface force density is displayed on the beam elements that are automatically created on the outer circumference of the calculation area during an analysis.
- Specify settings for [Add a layer of air in the computational domain force and torque] in mesh properties, when the (FEM + BEM) method is selected as the calculation method for a 2D analysis.

  Reference

  For more information, see "9-2. Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (JMAG-Designer)" on page 767.
- In a 3D analysis, if the surface force method is selected, shell elements are automatically created on the outer circumference of the calculation area during an analysis. The surface force density is displayed on those shell elements.
- When magnetizing properties are defined using the subroutine 'magusr', the subroutine 'usrene' needs to be defined as well, otherwise the electromagnetic force acting on the parts that have the magnetizing properties are not calculated.
- If the rotation motion condition and the electromagnetic force and torque calculation condition are specified on the same area (Fig. A) in a 2D analysis using the adaptive mesh option, JMAG automatically expands the calculation area of the electromagnetic force and torque calculation condition (Fig. A') to improve the calculation accuracy.



: region with the rotation motion condition set

: region with the electromagnetic force and torque calculation condition set

In this case, the torque and electromagnetic force for both before and after the expansion (A and A') are exported to a result file.
A file (e.g. AAA_input.jcf) which is the output file (e.g. AAA.jcf) containing mesh data is exported. The exported file contains the electromagnetic force and torque calculation condition set for the expanded calculation area.

- JMAG automatically extends the calculation area specified for the electromagnetic force and torque calculation condition to improve the accuracy of calculation even if the calculation area specified for the rotation motion condition and electromagnetic force and torque calculation condition are exactly the same in the analysis that takes into account eccentricity. However, the area where the mesh is distorted to simulate the eccentricity is not included in the calculation area.

3-19-2-1. Calculation Method for Electromagnetic Force

The electromagnetic force can be calculated using the following three methods.

■ Nodal force method

The nodal force method is used to obtain a electromagnetic force distribution in the area where the electromagnetic force and torque calculation condition is set. The electromagnetic force for each node is output.

The calculation using the nodal force method provides sufficient accuracy regardless of the settings for the calculation area.

Note that when magnetic materials are in contact with each other, calculation on only one of them is not possible.

▶▶ Reference

For more information, see P.221 "3-19-2-2. Calculation Area".

■ Surface force method

The surface force method is used to specify an integral path and to calculate stress on the integral path. The way of setting an integral path is different between 2D and 3D analyses.

For a 3D analysis, set this condition for the face of a gap. The face is regarded as an integral path.

For a 2D analysis, set this condition for a part. Among the edges of a part, the edge that is not in contact with a periodic boundary is automatically regarded as an integral path.

The calculation using the surface force method may not provide sufficient accuracy depending on the settings for the calculation area.

▶▶ Reference
For more information, see P.221 "3-19-2-2. Calculation Area".

■ Lorentz force method

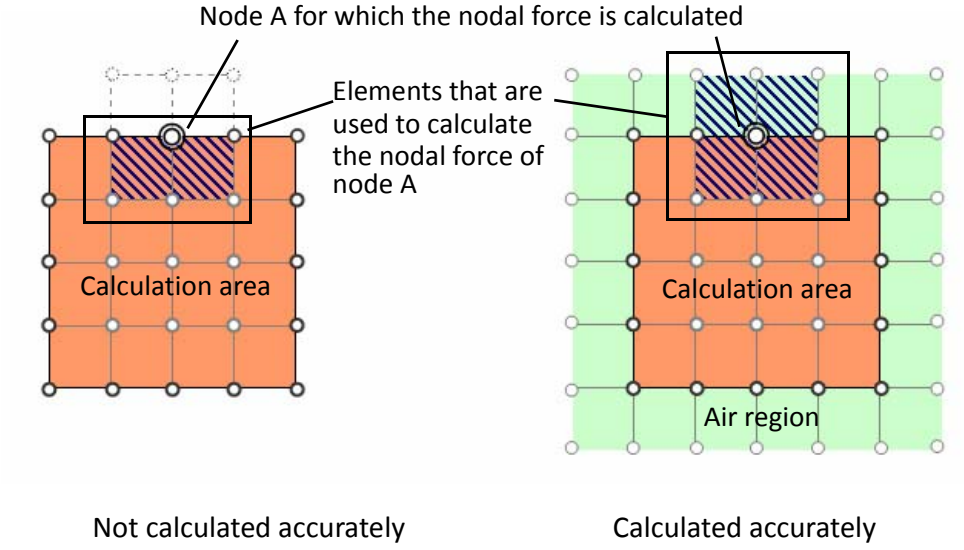
The Lorentz force method is used to obtain Lorentz force as electromagnetic force in all of the elements in the area where the electromagnetic force and torque calculation condition is set. When this method is used, the nodal force in the calculation area is not output.

Important **The Lorentz force is zero in the area where no current flows.**

3-19-2-2. Calculation Area

■ Nodal force method

- The force acting on each node on the outer circumference of the calculation area cannot be calculated accurately if there is no air region around the area where the electromagnetic force and torque calculation condition is set. This is because the nodal force is calculated for the elements that shares nodes.
  If the [FEM+BEM] method is selected in a 2D analysis, the air region needs not to be created around the model. Only when the nodal force method is used, the air region needs to be created around the area where the electromagnetic force and torque calculation condition is set. However, the air region needs not to be included in the calculation area of the condition.


Node A for which the nodal force is calculated
Elements that are used to calculate the nodal force of node A
Calculation area
Calculation area
Air region
Not calculated accurately
Calculated accurately

MEMO An air layer is automatically added in an area by specifying settings in [Add a layer of air in the computational domain force and torque], when (FEM + BEM) method is selected as the calculation method.

▶▶ Reference
For more information, see "9-2. Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (JMAG-Designer)" on page 767.

- If multiple magnetic materials are in contact with each other, create a thin air region between magnetic materials, and then specify the calculation area.

■ Surface force method

- In a 2D analysis, when calculating electromagnetic force acting on magnetic material that is placed in a space, it is necessary to create two air regions around the magnetic material, and then specify the magnetic material including the air regions as the calculation area.

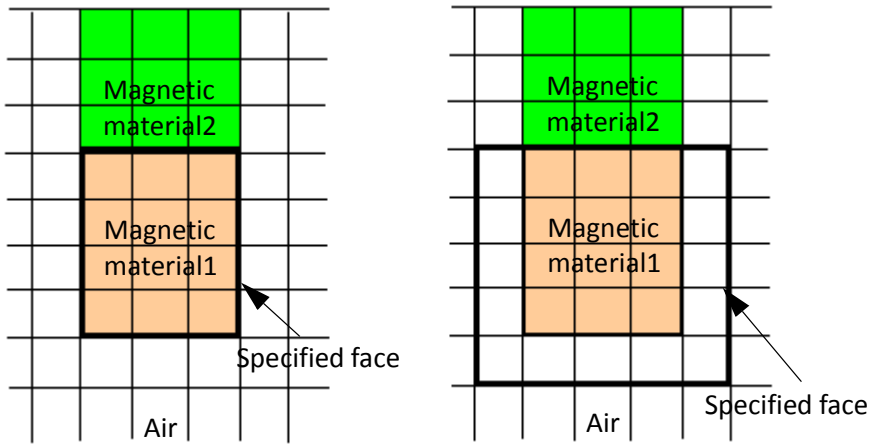
  MEMO An air layer is automatically modeled in an area by specifying settings in [Add a layer of air in the computational domain force and torque], when (FEM + BEM) method is selected as the calculation method.

  ▶▶ Reference
  For more information, see "9-2. Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (JMAG-Designer)" on page 767.

- In a 2D analysis, when the slide mesh (rotation, translation) option is not used, it is necessary to create an air region from the mover to the center of the gap, and then specify the mover including the air region as the calculation region.
  However, when the elements on the slide face may be distorted, the distorted elements should not be included in the calculation area.

  MEMO In a 2D analysis, if the slide mesh option is used, the area where the electromagnetic force and torque calculation condition is set is automatically expanded. In this case an air region is not necessary.

- In a 3D analysis, when calculating electromagnetic force acting on magnetic material that is placed in a space, if a face on the outer circumference of the magnetic material is selected, the electromagnetic force may not be calculated accurately.
  In this case, the face of the air region including the magnetic material needs to be selected. Air regions can be created as many as required.



(a) Bad example (b) Good Example

Method for selecting a calculation area to obtain the force acting on the magnetic material 1

- In a 3D analysis for a partial model, the periodic boundary face should not be specified as the calculation area for the electromagnetic force and torque calculation condition. If the electromagnetic force and torque calculation condition is set on a periodic boundary face, an integral path is set within a mover, and the calculation may not be accurate.

3-19-2-3. Linking to an External Circuit Simulator

The link to an external circuit simulator can be set in the transient response analysis of JMAG-Studio.

When the [Coupling with External Circuit Simulator] check box is selected, the electromagnetic force or torque can be output terminals to an external circuit simulator that is linked to JMAG. When an external circuit link condition is set, the X-, Y-, Z-direction component, and torque are displayed in the list of the [External Circuit Link] dialog box. Any of them can be selected as the output terminal.

MEMO For JMAG-Designer, if a force condition or torque condition is set, the force and torque that are obtained are automatically listed in the input/output terminals for an external circuit simulator.

3-20. Heat Source Calculation

▶▶ Reference

- For more information, see "Heat Source" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Heat Source" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-20-1. Function

The heat source condition is used for a coupled magnetic field analysis and thermal analysis. The data of heat generated in an FEM coil or in a resistance component obtained from the magnetic field analysis is used for the thermal analysis.

■ Related items

- Thermal analysis, Heat source condition

■ Targets

- Reference FEM coil condition
- Reference resistance component

■ Available modules

- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

■ Output items

- Heat source (W)

3-20-2. Important Notes Regarding the Use of This Function

- The heat source condition needs to be set when the heat source type is specified to the total amount for the heat source condition in thermal analysis.
- If the heat source condition is set for a partial model, the conversion factor needs to be set for both the FEM model and the circuit. The conversion factor can be specified with the analysis control condition in JMAG-Studio and with the study properties in JMAG-Designer.
- This condition is not required to run an independent magnetic field analysis.

3-21. Iron Loss Calculation

▶▶ Reference

- For more information, see "Iron Loss" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Iron Loss Calculation" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.
- See P.872 "12-3. Calculate Iron Loss".

3-21-1. Function

Execute magnetic field analysis and iron loss calculation continuously by setting iron loss as a condition. As for the reference result file referred to in the iron loss calculation, use the magnetic field analysis that was calculated previously.

Based on the data of magnetic flux density distribution contained in the reference result file, calculate the loss density distribution and loss amount in the specified frequency. It is possible to make evaluation by separating the iron loss into hysteresis loss and Joule loss.

■ Related items

- Material properties
- Stress distribution
- Tool: Iron loss calculation

■ Settings

- Basic frequency
- Stress
- Material type
- Components of magnetic flux density to reference
- Formula to calculate iron loss
- Calculation method of each component
- Conversion of coordinates system
- Calculation method of the required magnetic flux density at the time of calculating hysteresis loss

- Calculation method of the required magnetic flux density at the time of calculating Joule loss
- Start step and end step to refer
- Frequency scale factor * JMAG-Studio only
- Periodicity of the time series data of magnetic flux density
- Output of the iron loss of each step

MEMO In JMAG-Designer, some of the above settings can be set in the [Loss] tab in the [Material Editor] dialog box or the material properties for each part. For more information, see the JMAG-Designer User's Manual.

■ Available modules

- All magnetic field analysis modules

■ Output items

- Reference flux density (T)
- Iron loss density (W/m^3)
- Hysteresis loss density (W/m^3)
- Joule loss density (W/m^3)
- Iron loss (W)
- Hysteresis loss (W)
- Joule loss (W)
- Iron loss density (each component) (W/m^3)
- Hysteresis loss density (each component) (W/m^3)
- Joule loss density (each component) (W/m^3)
- Iron loss (each component) (W)
- Hysteresis loss (each component) (W)
- Joule loss (each component) (W)

3-21-2. Important Notes Regarding the Use of This Function

- The magnetic field analysis result data to be used in the iron loss analysis needs to include the following items.
  - Magnetic flux density
  - Displacement
- The iron loss analysis evaluates the flux density obtained from the magnetic field analysis and calculates the iron loss for each part with reference to the iron loss curve based on the flux density and frequency.
- The iron loss, hysteresis loss, and Joule loss are converted according to the specified conversion factor for the magnetic field analysis results.
- The following are the differences between specifying the Iron Loss condition for the magnetic field analysis in JMAG-Studio and running the iron loss calculation from the [Tool] menu.
  - In case iron loss is used as a condition, input cannot be made to the [Reference plot file] text box, and [Output file] text box with the [Iron loss Calculation] dialog box.

    This is because the name of the result file output after conducting calculation on magnetic field analysis JCF file to which iron loss conditions are set will be input to the [Reference plot file] text box. Furthermore, the output file name will be created automatically under the name of "(name of PLOT file of magnetic field analysis)_loss.plot".
  - In case iron loss is used as a condition in JMAG-Studio, there are no buttons to execute iron loss calculation in the [Iron loss Calculation] dialog box. This is because the iron loss calculation will be conducted at the time the calculation of the project to which the iron loss conditions are set, is executed.

Reference

As for other points, the specifications will be the same as the iron loss calculations of the tool. Verify the details with P.872 "12-3. Calculate Iron Loss".

- When iron loss calculation is set as a condition, the analysis will be run with the following flow. On this occasion, the magnetic field analysis data to which the iron loss conditions are set is referred to as “A”.

  i. Calculation of magnetic field analysis “A” will be executed.

  ii. The result file of the magnetic field analysis A is generated (For JMAG-Studio, A.plot, for JMAG-Designer, A.jplot)

  iii. The iron loss calculation begins with reference to the result file.

  iv. The iron loss calculation result is obtained; for JMAG-Studio, A_loss.plot; for JMAG-Designer, A_loss.jplot.

3-22. Magnetic Flux Calculation

▶▶ Reference

- For more information, see "Magnetic Flux Calculation" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG -Studio User's Manual.
- For more information, see "Magnetic Flux" in Chapter 29 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-22-1. Function

The magnetic flux condition is used to calculate the total amount of magnetic flux penetrating the specified cross section.

■ Related items

- N/A

■ Settings

- Regions to calculate magnetic flux

■ Available modules

- All of the modules of magnetic field analysis

■ Output items

- Magnetic flux (Wb)

3-22-2. Important Notes Regarding the Use of This Function

- Multiple groups can be set.

- When selecting an element (part) face, specify one of two element (part) faces as a cross section.

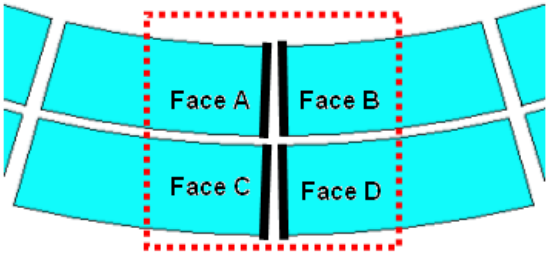


Fig.1. Enlarged view of a ring-shaped coil

As shown in Fig.1, if the area surrounded by red dotted lines is selected by dragging the mouse, both planes (plane A and plane B, plane C and plane D) in which the elements (parts) are in contact are selected.

As a result, accurate calculation results cannot be obtained.

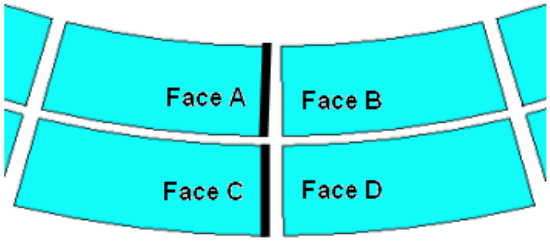


Fig.2 Enlarged view of a ring-shaped coil

As shown in Fig 2, select only planes on one side (plane A and plane C) of the area in which the elements (parts) are in contact.

3-23. Magnetostriction Calculation (JMAG-Designer)

▶▶ Reference
For more information, see "Chapter 29. Study Properties and Conditions of Magnetic Field Analysis," "Magnetostriction" in the "JMAG-Designer User's Manual."

3-23-1. Function

Calculates magnetostrictive force. The calculated magnetostrictive force can be used in a structural analysis.

■ Related items

- Materials
- One way coupled analysis for magnetic field and structure.
  - Structural analysis of electromagnetic force condition
  - Tool: Electromagnetic force settings for structural files
  - Tool: Electromagnetic force settings for Abaqus files

■ Setting

- Step * Frequency response analysis
- Number of divisions * Frequency response analysis

■ Modules that can be used with this function

- Entire modules

■ Output item

- Magnetostriction

3-23-2. Important Notes Regarding This Function

- Magnetostriction calculation for 3D shell is not supported.
- The magnetostrictive force is outputted as a time dependency value.
  Therefore, the magnetostrictive force calculated by a frequency dependency needs to be set so that it is converted to a time dependency in a frequency response analysis.
  Specify step to calculate magnetostrictive force and the number of divisions for frequencies in a period.
- The calculation result for magnetostriction is exported to a JPLOT file for specific usage. The file name is '(analysis title)_strain.jplot.'
- The following settings are required when calculating magnetostrictive force.
  - Specifying magnetostriction.
  - Specifying mechanical properties.
    - Magnetostriction properties (isotropic/anisotropy)
    - Mechanical properties (isotropic/anisotropy)
    - Poisson's ratio
    - Young's modulus
    - Shear modulus
- Specify a calculation for magnetostriction, in material properties for each part.
  Lamination direction also needs to be specified when setting anisotropic magnetostriction properties for material characteristics.

▶▶ Reference

See following chapters for details.

- "2-6-3. Specifying the Magnetostriction (Magnetic Field Analysis)" on page 38.
- "2-6-4. Specifying Mechanical Properties (Isotropic/Anisotropy) (Magnetic Field Analysis)" on page 39
- "3-5-1. Calculating Magnetostrictive Force" on page 97
- "3-5-2. Specifying Lamination Direction for Magnetic Steel" on page 97

- The magnetostriction calculated from a magnetic field analysis can be used in a structural analysis by the following methods.
  - Specify the result file for a magnetostrictive force, as a result file for a referenced magnetic field analysis, using the force condition of a structural analysis.
  - Specify the result file for a magnetostrictive force, as a result file for a referenced magnetic field analysis in the [Force Mapping Tool for Structural Analysis] tool.
  - Specify the result file for a magnetostrictive force, as a result file for a referenced magnetic field analysis in the [Force Mapping Tool for Abaqus] tool.
  - Specify a result file of a magnetostrictive force to export as a Nastran file format, in a output tool for a multi-purpose file.

Reference

See following chapters for details.

- "10-6-1. Magnetic Field Analysis (Static) => Structural Analysis (Static), Magnetic Field Analysis (Transient) => Structural Analysis (Static/Frequency)" on page 802
- "10-6-2. Magnetic Field Analysis (Frequency) -> Structural Analysis (Static/Frequency)" on page 807
- "12-9. Force Mapping for Structural Analysis" on page 954
- "12-11. Exporting Multi-purpose Files (JMAG-Designer)" on page 969
- "11-2-1. JMAG Magnetic Field Analysis (Force) > Abaqus Structural Analysis" on page 825

3-24. Partial Model

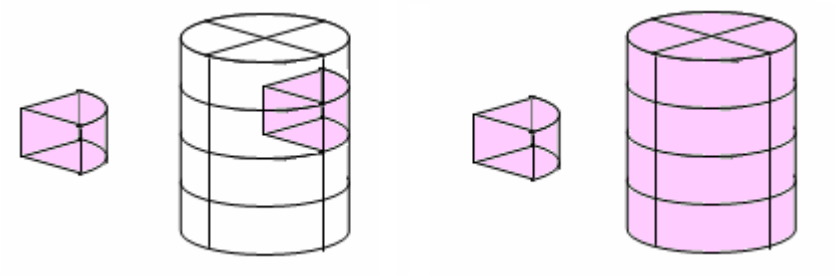
▶▶ Reference

- For more information, see “Partial Model” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Partial Model” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-24-1. Function

The partial model condition is used to set the parameters for mapping a partial model to a full model. This condition is used for a coupled magnetic field and thermal analysis. The heat source distribution can be mapped from a magnetic field analysis model to a thermal analysis model if the magnetic field analysis model is a smaller fraction of the thermal analysis model.

For example, suppose you want to map the heat source distribution obtained from a magnetic field analysis with a 1/4 partial model in the circumferential and axial directions to the full model of a thermal analysis.



Left: Without a partial model condition
Right: With a partial model condition (rotation/translation)

If a partial model condition is not set, the heat source distribution of a partial model is only mapped to the corresponding part of a full model. With a partial model condition, the heat source distribution of a partial model is mapped to a full model as shown in the figure above.

■ Related items

- Stress Distribution
- Coupled magnetic field and thermal analysis

■ Settings

- Periodicity/symmetry settings for the cross section

  Rotational periodicity:
  - Periodic boundary face
  - Periodic angle
  - Axis direction

  Translational periodicity:
  - Periodic boundary face
  - Periodic distance
  - Direction

  Mirror:
  - Boundary for the mirror plane

■ Available modules

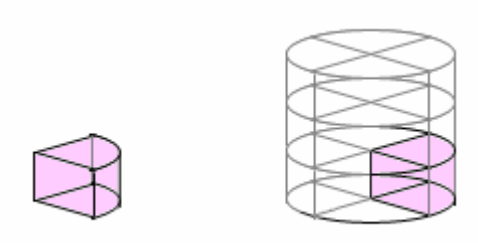
- All of the modules of magnetic field analysis

■ Output items

- N/A

3-24-2. Important Notes Regarding the Use of This Function

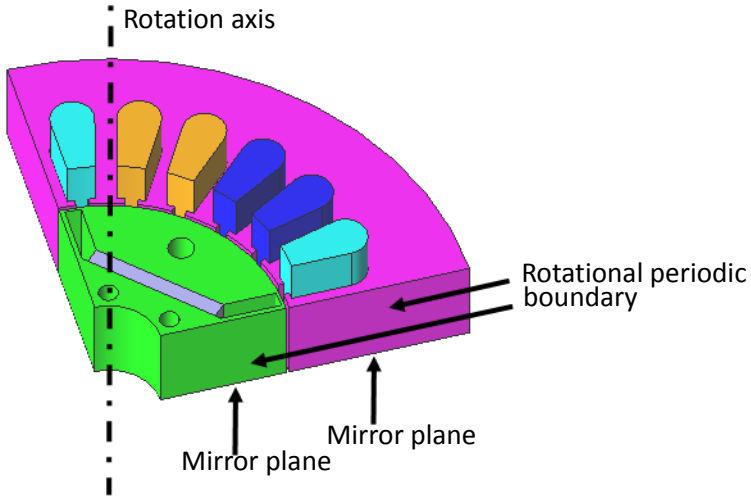
- If you specify both the rotation periodicity and translation periodicity, the partial model is copied in the axis direction.



Example

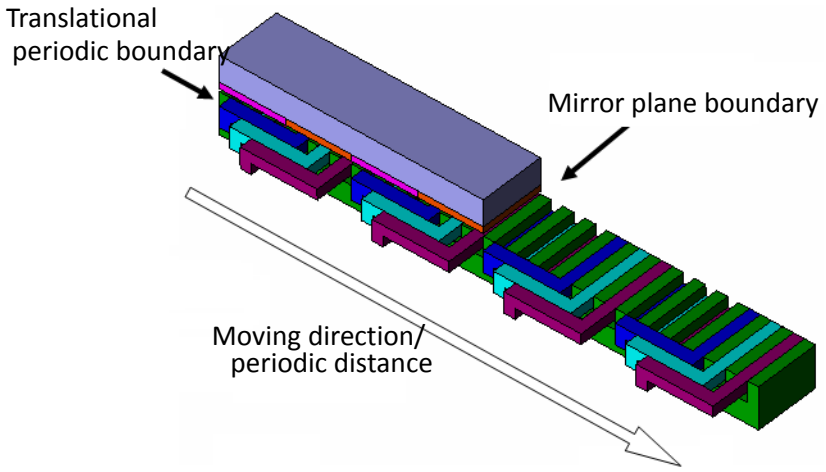
■ Example: Rotation motor

The figure below is a 1/8 model of a motor. [Rotational Periodicity] (periodic angle of 90 degrees) and [Mirror] are set on the cross section.



■ Example: Linear motor

The figure below is a 1/2 model of a linear motor. [Translational Periodicity] (the periodic distance is the distance modeled in the moving direction) and [Mirror] are set on the cross section.



3-25. External Circuit Link (Simulink, PSIM)

▶▶ Reference

- For more information, see "External Circuit Link" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see the "External Circuit Link 1&2" sections of Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-25-1. Function

It is possible to link JMAG to circuit/control simulators.

In this section, an explanation is given of the analysis procedure that couples with the two types of external circuit simulators, Simulink and PSIM (MagCoupler and MagCoupler-DL).

■ Related items

- Step
- External Circuit
- Subcycling
- Electromagnetic Force and Torque Calculation
- Motion

■ Settings

- Simulator to be linked
- Flag to use subcycling condition in the link
- Component name and terminal name on the external circuit simulator
- Relation between component and conditions

■ Available modules

- Transient response analysis (TR, DP)

3-25-2. Important Notes Regarding the Use of This Function

- The JMAG input file (*.jcf) needs to include data with the mesh generated regardless of the type of circuit/control simulator.

  Data with the following settings specified cannot be used to link to a circuit/control simulator when exporting input files because the data is exported as region or solid data rather than data with the mesh generated.

  - Generate mesh while running analysis (JMAG-Studio)
  - Layer mesh (JMAG-Designer)
  - Morphing (JMAG-Designer)
  - Adaptive mesh
  - Patch mesh
    - A motion condition is set in JMAG-Studio, [Standard Mesh] is selected under [Mesh Type] in the [Automatic Generation] dialog box, or the [Generate Mesh While Running Analysis] check box is selected.
    - A motion condition is specified in JMAG-Designer and the [Remesh Model at Each Step] check box is selected in the [Mesh Properties] dialog box.

3-25-3. Linking to Simulink

The magnetic field analysis solver can be linked to Simulink.

3-25-3-1. Required Module and Software

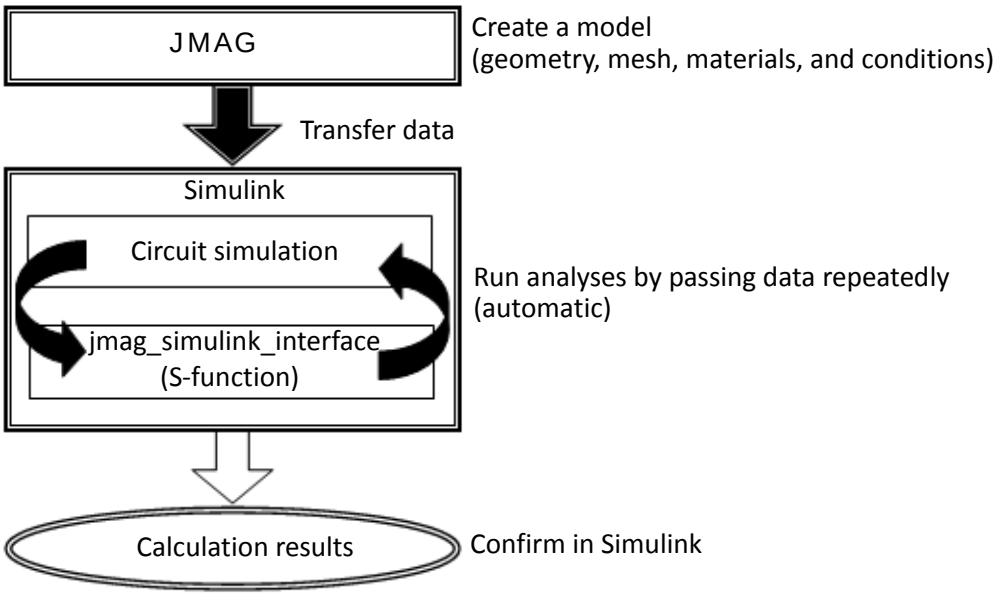
The following module and software are required.

- JMAG-Studio or JMAG-Designer
- Transient response analysis solver (TR or DP)
- MATLAB/Simulink (For 32 bit, R2006b or later. For 64 bit, R2007b or later.)

3-25-3-2. Analysis Flow

A dynamic data link can be established between Simulink and the JMAG magnetic field analysis solver using the data created with JMAG.

■ **Link to Simulink using S-function (by setting an external circuit link condition)**



3-25-3-3. Analysis with Simulink

An analysis with Simulink can be performed as follows.

1. Place the following files in the same directory.

- JMAG_SF.mdl
- icudt20.dll
- icuuc20.dll (32 bit) / icuuc20_64.dll (64 bit)
- xerces-c_1_6_0.dll (32 bit) / xerces-c_1_6_0_64_05.dll (64 bit)
- DFORMD.dll
- jmag_simulink_interface.dll (32 bit) / jmag_simulink_interface. mexw64 (64 bit)
- JCF file (e.g. jinput.jcf)
- XML file (e.g. jinput_csl.xml)

The MDL and DLL files are located in the JMAG installation directory. You can copy these files to use.

The XML file is exported at the same time as you export the data that contains the external circuit condition to a JCF file.

MEMO When an environment variable is specified, you can run an analysis with the four files; JMAG_SF.mdl, jmag_simulink_interface.dll, JCF file, and XML file placed in the same directory. For information about setting the environmental variable, see MEMO later in this section.

2. Start MATLAB/Simulink.

3. Open "JMAG_SF.mdl."

4. Copy all of the blocks from "JMAG_SF.mdl" to Simulink, or create a Simulink circuit in "JMAG_SF.mdl."

For information about creating a circuit for Simulink, see the Simulink manual.

5. Run an analysis after setting the four types of parameters that are required for analysis with Simulink.

The JMAG result file is exported.

MEMO The procedures to add the JMAG installation directory and JMAG installation directory\Solver\mod \mag\mod to the path for the environment variable.
(The default directory is C: \Program Files\JMAG-Designer (or JMAG-Studio) and C: \ProgramFiles\JMAG-Designer (or JMAG-Studio)\Solver\mod\mag\mod)

WindowsXP:
1. Select [Start] > [My Computer].
2. Click [Control Panel].
3. Double-click [System].
4. Click the [Advanced] tab, and then click [Environment Variables].
5. Select a path under [System variables], and then click [Edit] to open the [Edit System Variable] dialog box.
6. Add the JMAG installation directory and JMAG install directory \Solver\mod\mag\mod to the [Variable value] box.
(The default directory is C: \Program Files\JMAG-Studio (or JMAG-Designer) and C: \ProgramFiles\JMAG-Studio (or JMAG-Designer)\Solver\mod\mag\mod)
7. Click [OK], and close all of the dialog boxes.

Windows2000:
1. Select [Start] > [My Computer].
2. Click [Control Panel].
3. Double-click [System].
4. Click the [Advanced] tab, and then click [Environment Variables].
5. Select a path under [System variables], and then click [Edit] to open the [Edit System Variable] dialog box.
6. Add the JMAG installation directory and JMAG installation directory \Solver\mod\mag \mod to the [Variable Value] box.
(The default directory is C: \Program Files\JMAG-Studio (or JMAG-Designer) and C: \ProgramFiles\JMAG-Studio (or JMAG-Designer)\Solver\mod\mag\mod)
7. Click [OK], and close all of the dialog boxes.

3-25-4. Linking to PSIM

The magnetic field analysis solver can be linked to PSIM.

3-25-4-1. Required Module and Software

The following module and software are required to link to PSIM.

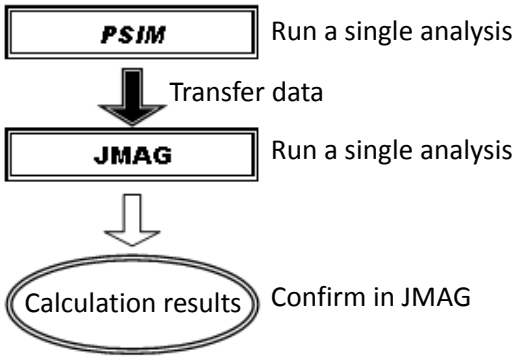
- JMAG-Studio or JMAG-Designer
- Transient response analysis solver (TR or DP)
- PSIM MagCoupler (7.1.2 or later versions) or PSIM MagCoupler-DL (7.1.2 or later versions)
  - PSIM 8.0.7 and earlier versions are supported by JMAG-Designer 10.1.1 or earlier versions, or JMAG-Studio 10.01301 or earlier versions.
  - PSIM 9.0.1 and later versions are supported by JMAG-Designer 10.1.2 or later versions, or JMAG-Studio 10.01401 or later versions.

3-25-4-2. Analysis Flow

There are two ways to link to PSIM; using the analysis results of PSIM in JMAG, or establishing a link between PSIM and the JMAG magnetic field analysis solver using the data created with JMAG.

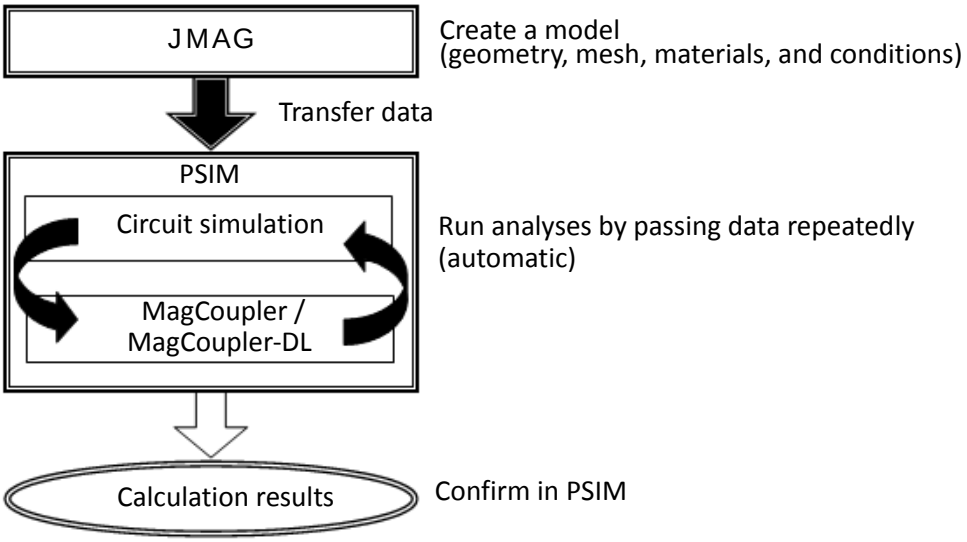
■ Using the analysis results of PSIM in JMAG

The flow of analysis is as follows.



■ Linking to PSIM and MagCoupler (by setting an external circuit link condition)

The flow of analysis is as follows.



MagCoupler and MagCoupler-DL are an interface for the analysis between JMAG and PSIM.

MagCoupler is used only when an external circuit link condition is set.

MagCoupler-DL is used when an external circuit link condition and subcycling condition are set.

MagCoupler can be used for the analysis with a diode, FEM conductor, reference FEM coil, and reference FEM conductor components that cannot be used if an subcycling condition is set. MagCoupler can also be used for the analysis with a switch component and the analysis that takes into account eddy currents.

On the other hand, MagCoupler-DL is a more stable interface as it does not have the variation of electric potential possibly caused by the switch timing of the circuit.

▶▶ Reference

For information about MagCoupler and MagCoupler-DL, see P.246 "3-26. Subcycling".

3-25-4-3. Analysis with PSIM Using the Dynamic Data Link

JMAG can be linked to PSIM as follows.

1. **Locate the JCF file and XML file exported from JMAG in the folder to use for the analysis with PSIM.**
2. **Start PSIM.**
3. **Select [Elements] > [Power] > [MagCoupler Module] > [MagCoupler], and then create a model.**
4. **Click the MAGCoupler block, and then select the XML file from [Link table file].**
5. **Run the analysis.**

   The analysis results of PSIM are displayed.

   The result file of JMAG are exported to the folder used for the analysis with PSIM.

MEMO The path to the JMAG installation directory needs to be specified in PSIM. The procedures are as follows.

1. In PSIM, select [Option] > [Set Path]. The [Set Path] dialog box is displayed.
2. In the [PSIM Search Path] box, enter the path to the JMAG installation directory and the JMAG installation directory \Solver\mod\mag\mod.

3-26. Subcycling

▶▶ Reference

- For more information, see "Subcycling" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see the "External Circuit Link 1&2" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-26-1. Function

The subcycling condition is used to set the time step for a magnetic field analysis when JMAG is linked to an external circuit simulator.

The analysis time may be extremely long if a magnetic field analysis is run for each step of an external circuit simulator. In this case, the time required for the simulation can be reduced by skipping the magnetic field analysis by setting the subcycling condition. The magnetic field analysis can also be set to skip if the current value or displacement is small.

In addition, the subcycling condition is used to reduce the time to link to an external circuit simulator when an external circuit link condition is set. An external circuit link condition needs to be set before setting a subcycling condition.

■ Related items

- External Circuit Link
- Electromagnetic Force and Torque Calculation
- Motion

■ Settings

- Time unit
- Rated current
- Flag to skip magnetic field analysis if the change is minimal
- Current threshold
- Motion type (rotation/translation)
- Displacement threshold
- Electromagnetic force and torque calculation condition and motion condition to link

■ Available modules

- Transient response analysis (TR, DP)

3-26-2. Important Notes Regarding the Use of This Function

■ Time Unit

Sets the timing of running an analysis in a magnetic field analysis. The timing of circuit calculation needs to be set in an external circuit simulator.

■ Rated Current

[Rated Current] is used to improve accuracy when there is a small excitation current.

JMAG runs a process to improve the accuracy if the excitation current is relatively small based on the [Rated Current] setting.

JMAG automatically determines whether or not to improve accuracy when [Rated Current] is not set (0A).

[Rated Current] is not normally specified, but the value can be changed if the results are abnormal, such as an inconsistent waveform.

■ Current Threshold

Sets the threshold for current variation to skip the magnetic field analysis if the amount of current is small. If the following formula is satisfied, the magnetic field analysis is run.

$$\mathrm{abs}(\mathrm{I}_{\mathrm{old}}) \times \mathrm{I}_{\mathrm{ratio}} \times 1/100 < \mathrm{abs}(\mathrm{I}_{\mathrm{now}} - \mathrm{I}_{\mathrm{old}})$$

I_{ratio} : Current threshold (%)

I_{old} : Current value when the previous magnetic field analysis is run

I_{now} : Current value at present

■ Displacement Threshold

Sets the threshold for displacement to skip the magnetic field analysis if the displacement of the mover is small. If the following formula is satisfied, the magnetic field analysis is run.

$$P_{threshold} < abs(P_{now} - P_{old})$$

$P_{threshold}$: Displacement threshold

(For rotational motion, the unit of measurement is (deg). For translational motion, the unit of measurement is (m).)

P_{old} : Position when the previous magnetic field analysis is run

P_{now} : Present position

MEMO The displacement threshold needs not to be specified for a model such as reactor that does not move.

■ Motion condition and electromagnetic force condition to link

When a subcycling condition is set, a magnetic field analysis is run to calculate inductance. Therefore, a motion condition, electromagnetic force, and torque condition need to be related using the subcycling condition to calculate electromagnetic force and torque during a circuit control simulation.

A motion condition, electromagnetic force condition, and torque condition need to be paired when linking to an external circuit simulator. Multiple pairs of these conditions can be set.

3-26-3. Analysis Flow Using the Subcycling Condition

3-26-3-1. PSIM

An analysis with PSIM is run as follows.

1. Set an external circuit link condition and a subcycling condition

Set the external port to connect to the external circuit simulator. Set the parameters for the subcycling condition such as the time step and threshold value to skip the analysis.

Important **A diode, switch, reference FEM coil, and reference FEM conductor components cannot be used in the circuit of JMAG if a subcycling condition is set when JMAG is linked to PSIM.**

MEMO The terminal that is specified by the electric potential source (1 terminal), current source, or electric potential source (2 terminals) component of JMAG is used as an interface of a JMAG model (external port) in PSIM when a subcycling condition is set. The terminal to be an external port needs to be specified for the electric potential source (1 terminal), current source, or electric potential source (2 terminals) component in the circuit of JMAG in advance.

2. Export a JCF file.

When a JCF file with the external circuit link condition and the subcycling condition set is exported, an XML file is also exported. This XML file is used to specify a JMAG model in PSIM.

MEMO The JMAG input file (*.jcf) needs to include data with the mesh generated regardless of the type of circuit/control simulator.

3. Set parameters in PSIM.

The MagCoupler-DL block is required to run a simulation in PSIM using the subcycling condition, specify the XML file exported from JMAG, and then specify each of the parameters.

▶▶ Reference

For information about the MagCoupler-DL block, see the PSIM manual.

4. Running an analysis in PSIM.

3-26-3-2. Simulink

When a subcycling condition is set, the link to Simulink can be established by following the procedures in P.241 "3-25-3-3. Analysis with Simulink".

Important **The FEM coil component and FEM conductor component cannot be used in the circuit of JMAG when a subcycling condition is used for the link to Simulink.**

3-27. Frozen Permeability (JMAG-Designer)

▶▶ Reference

- For more information, see “Magnetic Field Analysis Using the Frozen Permeability Condition” in Chapter 26 “Creating a Study by Selecting an Analysis Type” in the JMAG-Designer User's Manual.
- For more information, see “Frozen Permeability” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-27-1. Function

This condition is used to define the fix the permeability of each part based on the existing analysis results and define the material properties from the magnetic flux density and the magnetization curve.

■ Related items

- Material properties: Magnetization properties of soft magnetic material (20 page)
- Post output (316 page)

■ Settings

- Magnetic field analysis results file to refer (*.jplot, *.plot)
- Use average values flag
- Setting Type

■ Available modules

- All magnetic field analysis modules

■ Output items

- N/A

3-27-2. Important Notes Regarding the Use of This Function

- Materials that have anisotropic magnetization properties or lamination cannot be used.
- The adaptive mesh feature, remesh model at each step feature, morphing feature, and the layer mesh feature cannot be used with the frozen permeability condition.
- Batch execution of analyses using the remote system is not supported.
- Two analyses are required when this condition is used. Furthermore, the geometry of the mesh in the second analysis needs to match the first analysis.
- The relative permeability and differential permeability need to be exported as results for the first analysis.

3-28. Stress Distribution

▶▶ Reference

- For more information, see “Stress Distribution” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Stress Distribution” in Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.
- For more information about this condition, see P.801 "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction".

3-28-1. Function

Sets the stress value. Two types are available: a method setting the stress value per material and a method setting the stress value by referring to the results of the structural analysis.

Result files exported from Abaqus can be specified.

▶▶ Reference

For more information about referring to Abaqus result files, see "Chapter 11. Coupled Analysis Using Abaqus" on page 823.

■ Related items

- Material properties: Magnetization properties of soft magnetic material (20 page)
- Iron loss calculation condition (226 page)
- Tool: Iron loss calculation (872 page)

■ Targets

- Entire model

■ Settings

- Setting stress values
  - One way coupled analysis (a result file for JMAG structural analysis or Abaqus structural analysis (*.jplot, *.plot, *.odb) is used)
  - Two-way coupled analysis (fully coupled analysis for thermal and stress)
  - Specify for each material

■ Available modules

- All of the modules of magnetic field analysis

■ Output items

- Magnetic flux density, induced voltage, etc. * In case of magnetic field analysis.
- Iron loss, etc. * In case of iron loss analysis based on iron loss calculation tool.

3-28-2. Important Notes Regarding the Use of This Function

- This condition is used when the material data has stress dependent magnetic property.

  ▶▶ Reference
  For more information, see P.29 "2-3-5. Specifying the Magnetic Properties of a Stress Dependency".

- Refer to the compressive stress for the magnetizing properties of stress dependence and iron loss curves.
- When a stress value is set for each material, specify a negative value (compressive stress value)
  Example) -10 MPa

3-29. Temperature Distribution (JMAG-Designer)

▶▶ Reference

- For more information, see “Chapter 29. Study Properties and Conditions of Magnetic Field Analysis,” “Temperature Distribution,” in the “JMAG-Designer User's Manual.”
- For more information, see P.769 "Chapter 10. Coupled Analysis".

3-29-1. Function

An analysis is run using materials with temperature dependency by referring to thermal analysis results.

■ Related items

- Material properties: Magnetization properties of soft magnetic material (20 page)
- Material properties: Electric properties (31 page)
- Two-way coupled analysis

■ Setting

- Coupling type
  - One way coupled analysis (Result file of JMAG thermal analysis (*.jplot, *.plot) is used)
  - Two-way coupled analysis (Abaqus fully coupled analysis for thermal and stress)
- Result file of thermal analysis for referencing (one-way)

■ Modules that can be used with this function

- The entire magnetic field analysis modules

■ Output item

- Magnetic flux density, induced voltage

3-29-2. Important Notes Regarding the Use of This Function

- This condition is specified when using material data with temperature-dependent magnetic property.

3-30. Displacement (JMAG-Designer)

▶▶ Reference

- For more information, see “Chapter 29. Study Properties and Conditions of Magnetic Field Analysis,” “Displacement,” in the “JMAG-Designer User's Manual.”
- For more information, see "Chapter 10. Coupled Analysis" on page 769.
- For more information, see "Chapter 11. Coupled Analysis Using Abaqus" on page 823.

3-30-1. Function

An analysis is run using displacement in a part by referring to the results of a structural analysis.

■ Related items

- Two-way coupled analysis

■ Setting

- Coupling type
  - One way (Result file of JMAG structural analysis or Abaqus structural analysis (*.jplot, *.plot, *.odb) is used)
  - Two way (Abaqus coupled structural analysis)
- Result file of structural analysis for referencing (one-way)

■ Modules that can be used with this function

- The entire magnetic field analysis modules

■ Output items

- Magnetic flux density, induced voltage

3-30-2. Important Notes Regarding the Use of This Function

- [Only remesh air region] can be selected for a two-way coupled analysis when this condition is selected.
- The model can run an analysis while a model is deformed from using the displacement function but parameters such as the magnetization direction cannot be modified.

3-31. Reusing Demagnetization (JMAG-Designer)

▶▶ Reference

For more information, see "Chapter 29. Study Properties and Conditions of Magnetic Field Analysis," "Reusing Demagnetization," in the "JMAG-Designer User's Manual."

3-31-1. Function

A demagnetized magnet used in another analysis can be reused as an initial value for a magnet used to run a new analysis.

■ Related items

- Material properties: Magnetization properties of permanent magnet (10 page)

■ Setting

- Result file for transient response analysis with data for demagnetized magnets.

■ Modules that can be used with this function

- Static analysis (ST)
- Transient response analysis (TR, DP)

■ Output item

- None

3-31-2. Important Notes Regarding This Function

- Information in the last step for irreversible demagnetization is obtained from a result for a transient response analysis using demagnetized magnets. This occurs in the reusing demagnetization condition where the information can be used as an initial value for a magnet when running a new analysis.
- The following demagnetized magnets can be reused.
  - Irreversible demagnetization magnet with demagnetizing field
    - Alnico magnets (irreversible) provided by Hitachi Metals for the material database
    - Custom materials ([Nonlinear (irreversible / Demagnetizing field)] is selected in [Magnetic Property Type] in material editor).
  - Irreversible demagnetization for temperature dependency with demagnetizing field
    - Neodymium sintered magnet (irreversible) and ferrite magnet (irreversible) provided by Hitachi Metals for the material database.
    - Custom materials ([Nonlinear (Irreversible / Thermal demagnetization / Demagnetizing field)] is selected in [Magnetic Property Type] in material editor).
- The following are restrictions when using reusable demagnetized magnets.
  - Irreversible demagnetized magnet described above.
  - Matching model geometries (dimension, periodicity).
    * Models with different mesh can run an analysis.
  - The same magnetic material is used for the analysis to obtain a specified JPLOT file and the analysis specifying the condition.
- An analysis reusing a demagnetized magnet is as follows.

  i. Running an analysis using demagnetized magnets.

  ii. Specifying the reusing demagnetization condition for a study running a new analysis.

  iii. Select a magnet in the reusing demagnetization condition, and then specify the JPLOT file obtained in the analysis from step i.

  iv. Running an analysis.

3-32. External Field

▶▶ Reference

- For more information, see "External Field" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "External Field" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-32-1. Function

Gives uniform external field overall the model.

■ Related items

- Symmetry boundary (108 page)

■ Settings

- Intensity of uniform external field (T): This is specified in any of the following three ways.
  - Point sequence data (ST, TR, FQ)

    Defines function by point sequence comprising of time or other parameters and values of magnetic flux density.
  - Sinusoidal function (TR, DP, FQ)

    The sinusoidal function is defined by amplitude *A*, frequency *F*, and phase *P*.

    $I(t) = A \cdot \sin(2\pi \cdot F + P \cdot \pi/180)$

    In the frequency response analysis, frequency can not be specified in supply current condition.
  - Exponential function (TR, DP)

    The exponential function is defined by asymptotic value C_0, amplitude *AMP*, and time constant *T*.

    $I(t) = C_0 + AMP \cdot \exp(-t/T)$
- Direction vector of uniform external field

■ Available modules

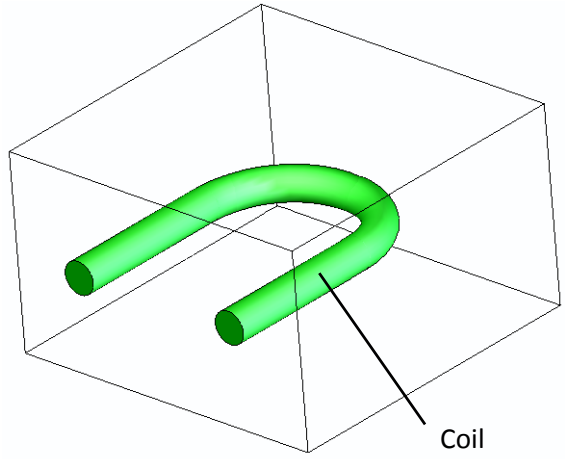
- 3D analysis

■ Output items

- Magnetic flux density (T)

3-32-2. Important Notes Regarding the Use of This Function

- For models in which cross sections of the magnet and conductor are located on the symmetry boundary plane, uniform external field must be given in parallel with the symmetry boundary conditions.
- The external field condition cannot be set on a model with a divided coil as shown below.



Example:
A model where the external field condition cannot be set (1/2 model)

3-33. Gap

▶▶ Reference

- For more information, see "Gap" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Gap" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-33-1. Function

Sets a very thin magnetic air gap on a face.

■ Related items

- Material properties: Gap element

■ Targets

- Element face group
- Solid face group

■ Settings

- Face to set the magnetic gap on
- Thickness of gap

■ Available modules

- 3D analysis

■ Output items

- Magnetic flux density (T)

3-33-2. Important Notes Regarding the Use of This Function

- In 3D analyses, shell elements are automatically generated on the face where this condition is set during an analysis. The magnetizing properties of the air are applied to those elements.
- Use this condition when the thickness does not need to be included in the calculation or the gap is too thin to be meshed.
- This condition should be set on a face through which a magnetic field passes longitudinally. Otherwise, the calculation results will be incorrect.
- The following conditions cannot be set on the face where this condition is set.
  - Symmetry Boundary
  - Current
  - Current Density
  - FEM Coil
  - FEM Conductor
  - Slide

3-34. Insulation

▶▶ Reference

- For more information, see "Insulation" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Insulation" in Chapter 29 "Study Properties and Conditions of Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-34-1. Function

The insulation condition is used to include the effect of electric insulation in the analysis where the effect of eddy currents are taken into account.

■ Related items

- Material properties: Electrical properties

■ Settings

- Insulated face

■ Available modules

- Transient response analysis (TR, DP) * Except axisymmetric
- Frequency response analysis (FQ) * Except axisymmetric

■ Output items

- Current density (A/m^2)
- Joule loss density (W/m^3)

3-34-2. Important Notes Regarding the Use of This Function

- This condition can be set for each material to take into account the effect of insulation.

  ▶▶ Reference

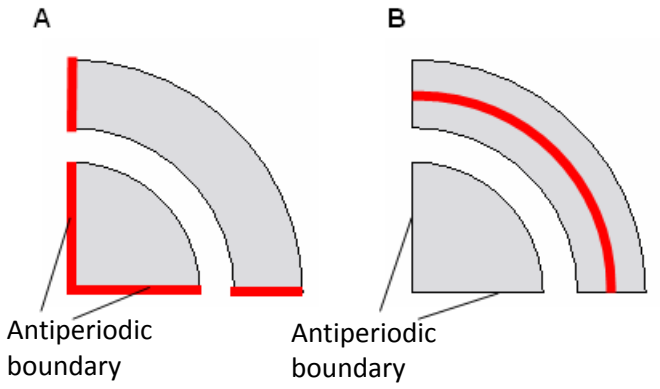
  For more information, see P.99 "3-6-2. Allowing Electric Insulation".

- This condition specifies a face to separate a conductor to take into account electric insulation. However, the mesh does not change before and after the analysis.
- The insulation condition can only used for a FEM conductor in a 2D analysis. This function has no effect on results when used with other conductors.
- The insulation condition cannot be set for an inflow face of a FEM conductor and a face that it is connected with for a 3D analysis.

  ▶▶ Reference

  For more information about FEM conductors, see "3-14. FEM Conductor" on page 141.

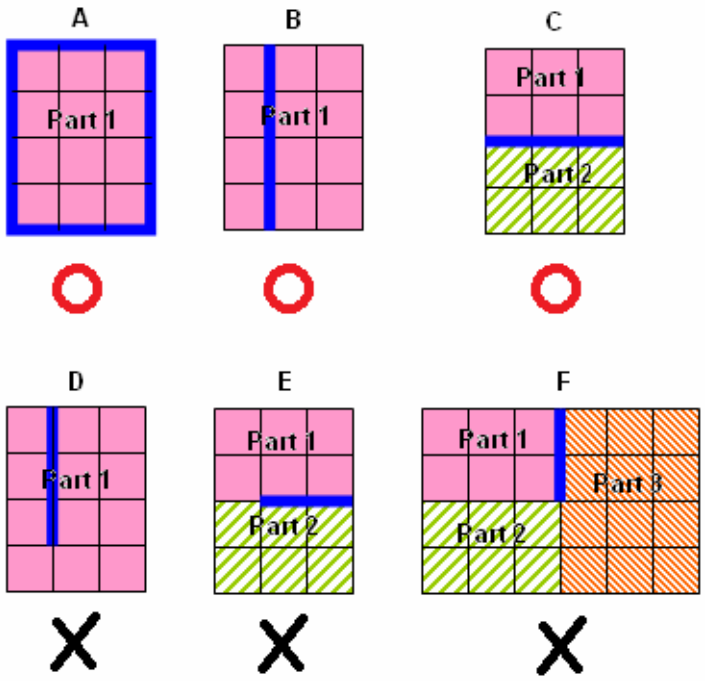
- This condition should not be set to an antiperiodic boundary face. For example, the red line displayed in figure A is set to the antiperiodic boundary condition (1st periodic boundary plane) and the 2nd periodic boundary plane based on the 1st periodic boundary plane and the periodic angle.
  This condition cannot also be set with a face intersecting an antiperiodic boundary face as shown in Fig. B.
  Extend the models from an antiperiodic boundary condition to a periodic boundary condition or a partial model to a full model, in order to set the insulation condition to the areas explained previously.



Areas that the insulation condition cannot be set

- This condition can be set for an area that is in contact with the end of a part or air.

  For example, when an insulation condition is set for a bold line in the mesh model A to F as shown in the figure below. The insulation condition is effective when the condition is set from one edge to another as shown in Fig. B and C. On the other hand, the insulation condition is not effective when the condition is set for a part partially as shown in Fig. D, and also when electric conductivity is set for parts that have contact.



Blue line: Insulation condition O: Correct setting X: Incorrect setting

3-35. Partial Inductance

▶▶ Reference

- For more information, see “Partial Inductance” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “JMAG-Designer User's Manual,” “Chapter 29 Study Properties and Conditions of Magnetic Field Analysis,” “Study Properties.”
- For more information, see P.998 "12-14. Calculate Busbar Inductance".

3-35-1. Function

In case the busbar inductance tool is used, specify the region that will be the object of calculation.

■ Related items

- Busbar inductance tool

■ Settings

- Region that will be the object of inductance calculation
- Plane that will be the inflow port of the current

■ Available modules

- Frequency response analysis (FQ(3D))
  * This cannot be handled in case of 2D analysis, axially symmetric analysis.

■ Output items

- Self inductance (H)
- Mutual inductance (H)
- Resistance (ohm)

3-35-2. Important Notes Regarding the Use of This Function

- In case partial inductance calculation tool is used, always set these conditions.
- The flow of inductance calculation is as follows:

  i. Start up the project of 3D frequency response analysis.

  ii. Conduct operations up to material setting (electric properties), condition setting (FEM conductor condition, partial inductance condition), and mesh creation.
  *** External circuits are not required.**

  iii. Export the created data to JCF file.

  iv. Start up the bus bar inductance tool.

  v. Specify the JCF file that was created by Procedure 3) as input file.

  vi. Specify parameters such as frequency.

  vii. Execute inductance calculation.

- Joint use of adaptive mesh and patch mesh (generate mesh while running analysis) cannot be done.
- Divide the materials per calculation region and specify a partial inductance condition for each region when multiple inductance calculation for a region exist in a part in JMAG-Studio.

3-36. Analysis Type

▶▶ Reference

For more information, see Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.

3-36-1. Function

Select the analysis type.

■ Setting

- Analysis type
  - Static analysis
  - Transient analysis
  - Frequency response analysis

3-36-2. Important Notes Regarding the Use of This Function

- Select an analysis type when creating a study for JMAG-Designer.
- Select an analysis type when creating a project file for JMAG-Studio.

3-37. Calculation Method

3-37-1. Function

In 2D analyses, one of the following calculation methods needs to be selected.

▶▶ Reference

- For more information, see "Analysis Control: Basic Settings" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 29 "Study Properties and Conditions of the Magnetic Field Analysis" in the JMAG-Designer User's Manual.

■ Direct method (FEM + BEM)

The boundary element method (BEM) is used. A solution can be obtained without being concerned about convergence. The calculation speed, however, is slow and a large amount of memory is required. This method is not appropriate for a large-scale model.

■ Iterative method (FEM)

The boundary element method (BEM) is not used. The calculation speed is faster than that for the direct method and less memory is required. A larger scale model can be used. However, highly accurate results may not be obtained for large-scale models or coarse mesh models because the calculation does not converge.

The difference of memory requirement and analysis time is indicated below:

Amount of memory used: FEM (iterative method) < FEM + BEM (direct method)

Analysis time: FEM (iterative method) < FEM + BEM (direct method)

3-37-2. Important Notes Regarding the Use of This Function

- In 3D analyses, the iterative method (not using BEM) is used. The direct method is not available.
- The following functions cannot be used when the direct method (FEM+BEM) is selected.
  - Slide mesh (rotation/translation)
  - Gap flux boundary condition
  - Reference FEM coil component
  - Parallel computing function (SMP, DMP)
  - Use GPU function
  - Time periodic explicit error correction function

3-38. Step Control

▶▶ Reference

- For more information, see "Study Properties" in Chapter 29 "Study Properties and Conditions of the Magnetic Field Analysis" in the JMAG-Designer User's Manual.
- For more information, see the section 'Step' in "Chapter 9. Setting Conditions of Magnetic Field Analysis," in the "JMAG-Studio User's Manual."

3-38-1. Function

The timing (time or position) when an analysis is run and obtaining results can be set for the variations in an analysis during calculation.

■ Related items

- Motion: Equation of motion
- External circuit link
- Output settings: Control of output interval
- User subroutine: usrstp

■ Setting

- Number of steps
  Specify the number of steps to run an analysis.
  - The range in analysis time is determined by the number of steps and time interval (or displacement) for a transient response analysis.
  - An independent analysis (transient is not used) is run for each step for a static analysis. Modify the number of steps when running multiple static analyses.

  MEMO The number of steps specified for a static analysis is only used in JMAG-Studio.

- Specifying analysis time
  The types of specifying method are as follows.
  - Regular interval/Time interval
    Specify steps in constant time intervals.
  - Time point sequence
    Method for specifying analysis time for each step using point sequence.
  - Regular intervals per section
    Method for specifying steps in constant time intervals for each section by dividing steps in multiple sections.
  - Displacement
    Method for specifying steps in constant displacement intervals. This function is used with the equation of motion.
  - Set by external program
    Method for specifying analysis time from an external program.

    ▶▶ Reference
    For more information, see "3-38-2-1. Specifying Method for Analysis Time" on page 274.

■ Modules that can be used with this function

- Static analysis (ST) *JMAG-Studio only
- Transient response analysis (TR, DP)

■ Output items

- Time

3-38-2. Important Notes Regarding This Function

- The step control is referred to other conditions such as rotation speed, current, time dependency of point sequence for circuit resistance, and step ID dependency. The area of analysis time defined by the step control needs to be set so that it does not exceed the area of time defined in other conditions.
- The steps which are outputted on a result file (*.jplot, *.plot) can be specified from the steps defined in the step control.

  ▶▶ Reference

  For more information, see "3-52-2-4. Controlling Output Intervals" on page 320.
- The step IDs and analysis time for each step can be changed by using a user subroutine *usrstp*(366 page).

3-38-2-1. Specifying Method for Analysis Time

This section describes each of the specifying methods for analysis time.

- Time interval
- Regular intervals
- Time point sequence
- Point sequence with regular intervals per section
- Displacement
- Set by External Program

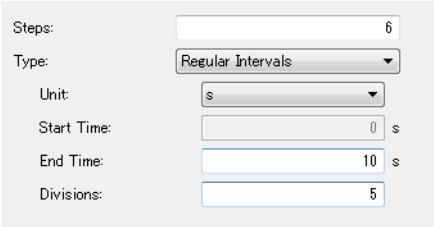
■ Time interval (JMAG-Designer)

- This setting is specified when positioning steps in a constant time interval from the first to the last step. Specify a time interval.

■ Regular intervals

- This setting is specified when positioning steps in a constant time interval from the first to the last step.
- The time interval is defined by specifying the start time of the analysis and the end time of the analysis, and the number of divisions defined in an area.
- The time for each step is defined by the following.
  Start time + (end time - start time) / number of divisions x (number of steps - 1)
- Each step that has exceeded the number of divisions from the defined time interval, are automatically set with a time, when the number of steps exceeds the number of division.

  Example) Specify the following settings to perform calculation every two seconds between 0 and 10 seconds.



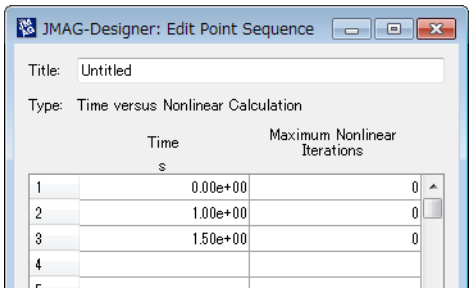
Regular intervals

The following example describes the time of the seventh step to be 12 seconds when [Steps] is specified to '7' (12 = 0 + (10 - 0) / 5 x (7 - 1)).

■ Time point sequence (unequally-spaced specification)

- This setting is selected when there is no constant interval in time for each step.
- Specify the time and nonlinear iterations for each step using point sequence.

  Example)



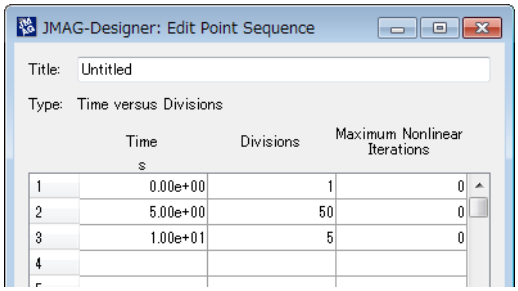
Time point sequence

  The time is specified for each step when point sequence is specified for an image similar to the one above.
  - Step 1 is 0 sec
  - Step 2 is 1 sec
  - Step 3 is 1.5 sec
- The steps defined in the point sequence is calculated and the analysis ends based on indefinite analysis time, regardless of the remaining steps (For this example, the analysis ends at step 3 regardless that the number of steps specified is 4).
- The value specified in [Maximum Nonlinear Iterations] of [Nonlinear Calculation] is used as the default value during an analysis when zero is specified for each step in maximum nonlinear iterations.

■ Point sequence with regular intervals per section

- This setting is selected when there are multiple sections in a step that are positioned in constant time interval for each section (multiple specification of 'regular interval').

  Example)



Point sequence with regular intervals per section

The time is specified for each step when point sequence is specified for an image similar to the one above.

* Start time is 0.0 sec

- Calculation is run in intervals of 0.1 second from 0 to 5 seconds (steps 1 to 51)
- Calculation is run in intervals of 1 second from 5 to 10 seconds (steps 52 to 56)
- Calculation is run in intervals of 1 second from steps 57 and after

- The value specified in [Maximum Nonlinear Iterations] of [Nonlinear Calculation] is used as the default value during an analysis when zero is specified for each step in maximum nonlinear iterations.

■ Displacement

- This setting is selected for specifying time in each step based on the displacement of 1 step.
- This function is used with the equation of motion for the motion condition.
  - Specify the motion condition and select [Equation of Motion] as the displacement type for JMAG-Designer.
  - Specify the motion condition and select [Equation of Motion] as the displacement type for JMAG-Studio. Select [Specify by Displacement] as the specifying method for the time and displacement of the equation of motion.
- Specifies the displacement for each step. The amount of time it takes to move the step based on displacement is automatically calculated to run the calculation.
- The time required to move a displacement for a step has no reference to the number of steps where the analysis ends when the time specified in [Maximum time interval] has exceeded.

■ Set by External Program

- Uses jbdll.dll from an external program during the analysis.

▶▶ Reference

For more information about jbdll.dll, see P.390 "3-57. DLL of Magnetic Field Analysis (jbdll.dll)".

3-39. Frequency Control

▶▶ Reference

- For more information, see "JMAG-Designer User's Manual," "Chapter 29 Study Properties and Conditions of Magnetic Field Analysis," "Study Properties."
- For more information, see the "Step" and "Frequency" sections of Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.

3-39-1. Function

The frequency when an analysis is run and obtaining results can be set for the variations in an analysis during calculation.

■ Related items

- Iron loss analysis
- Coupled analysis

■ Setting

- Number of steps
  Defines the number of frequencies to run an analysis. Modify the number of steps other than '1' when running a multiple frequency analyses.
- Frequency
  The types of specifying method are as follows.
  - Regular interval / Frequency interval
    Method for specifying steps in constant frequency interval.
  - Frequency point sequence
    Method for specifying each step and frequency using point sequence.
  - Regular intervals per section
    Method for specifying steps in constant frequency intervals for each section by dividing steps in multiple sections.
  - Time dependency of frequency point sequence
    Method for specifying frequency that varies with time. This setting is used when coupling with thermal analysis.

▶▶ Reference

For more information, see P.279 "3-39-2-1. Specifying Method for Analysis Frequency".

■ Modules that can be used with this function

- Frequency response analysis (FQ)

■ Output items

- Frequency

3-39-2. Important Notes Regarding This Function

- The frequency control refers to other conditions such as rotation speed, current, frequency dependency of point sequence for circuit resistance, and step ID dependency of point sequence. The area of analysis frequency defined in the frequency control needs to be set so that it does not exceed the area of frequencies defined by other conditions.

3-39-2-1. Specifying Method for Analysis Frequency

This section describes each of the specifying methods for analysis time.

- Frequency Interval (JMAG-Designer)
- Regular intervals
- Frequency point sequence
- Point sequence with regular intervals per section
- Time dependency of frequency point sequence (JMAG-Designer)

■ Frequency Interval (JMAG-Designer)

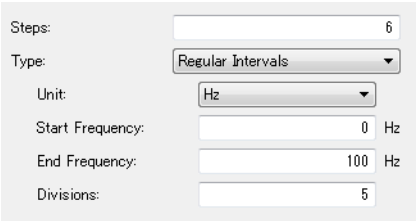
- This setting is specified when positioning steps in a constant frequency interval from the first to last step. Specify a frequency interval.

■ Regular intervals

- This setting is specified when positioning steps in a constant frequency interval from the first to last step.
- The frequency interval is defined by specifying the number of divisions determined from the initial value and end point of an analysis frequency.
- The frequency for each step is defined by the following.

  Initial value + (End point - initial value) / number of divisions x (number of steps)
- Each step that has exceeded the number of divisions from the defined frequency interval, are automatically set with a frequency, when the number of steps exceeds the number of division.

  Example) Specify the following settings to perform calculation every 20Hz between 0 and 100 Hz.



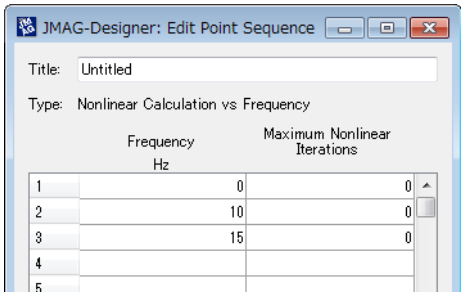
Regular intervals

The following example describes the frequency of the seventh step to be 120Hz when [Steps] is specified to '7' (120 = 0 - (100 - 0) / 5 × (7 - 1)).

■ Frequency point sequence (unequally-spaced specification)

- This setting is selected when there is no constant interval in a frequency for each step.
- Specify the time and nonlinear iterations for each step using point sequence.

  Example)



Frequency point sequence

The frequency is specified for each step when point sequence is specified for an image similar to the one above.

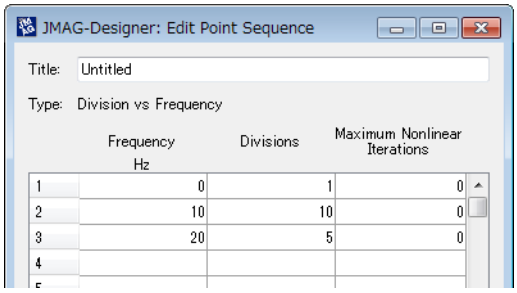
- Step 1 is 0Hz
- Step 2 is 10Hz
- Step 3 is 15Hz

- The steps defined in the point sequence is calculated and the analysis ends based on indefinite analysis frequency, regardless of the remaining steps (For this example, the analysis ends at step 3 regardless that the number of steps specified is 4).
- The value specified in [Maximum Nonlinear Iterations] of [Nonlinear Calculation] is used as the default value during an analysis when zero is specified for each step in maximum nonlinear iterations.

■ Point sequence with regular intervals per section

- This setting is selected when there are multiple sections in a step that are positioned in constant frequency interval for each section (multiple specification of 'regular interval').

  Example)



Point sequence with regular intervals per section

The frequency is specified for each step when point sequence is specified for an image similar to the one above.

* Initial value of frequency is 0.0Hz

- Calculation is run in intervals of 1Hz from 0Hz to 10Hz (steps 1 to 11)
- Calculation is run in intervals of 2Hz from 10Hz to 20Hz (steps 12 to 16)
- Calculation is run in intervals of 2Hz from steps 17 and after

- The value specified in [Maximum Nonlinear Iterations] of [Nonlinear Calculation] is used as the default value during an analysis when zero is specified for each step in maximum nonlinear iterations.

■ Time dependency of frequency point sequence (JMAG-Designer) * For coupled analysis

- This setting is selected when running a coupled magnetic field and thermal analysis for induction heating.
  Specifies the time and frequency using point sequence.
  The frequency for each step in a magnetic field analysis is determined during a coupled analysis. This is done by comparing the time defined in the step control for a thermal analysis and the point sequence defined in the magnetic field analysis.
- The calculation for the first step is run using the frequency defined in the first row of a point sequence. This setting is specified when a time dependency of frequency point sequence is selected for an analysis other than a coupled analysis.
- The time for the time dependency of frequency point sequence starts from 0 seconds, where the setting needs to be specified in ascending order.
- An analysis frequency is set with linear interpolation in the time defined in the time dependency of frequency point sequence.
- The frequency for the time defined in the maximum time in a point sequence is used, when the time defined outside the defined area in the time dependency of frequency point sequence is transferred from a step control in a thermal analysis.
- The specified number of steps is not used for a time dependency of frequency point sequence. A frequency analysis is run for the steps specified in a step control of a thermal analysis.

▶▶ Reference

For information about coupled analyses, see P.769 "Chapter 10. Coupled Analysis".

3-40. Full Model Conversion

▶▶ Reference

- For more information, see “Analysis Control: Basic Settings” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-40-1. Function

- When the full model conversion feature is not used, the values of a partial model are obtained. For example, when electromagnetic forces are calculated for a 1/4 model, the magnitude of forces acting on the model can be obtained. The values of a partial model must be multiplied by 4 to obtain the values of a full model.
  For 2D analyses, forces per meter in the Z-axis direction can be obtained.
  For axisymmetric analyses, forces per period can be obtained.
- When the full model conversion feature is used, the values of a full model are obtained. The multiplier is determined from the periodic angle when a periodic boundary condition is set.For example, the results are multiplied by 4 if the periodic angle is 90 degrees. In addition, if a multiplier other than periodic boundary is set, the results are also multiplied by the multiplier.
  For 2D analyses, the results are also multiplied by the thickness of the model.

■ Output items to be multiplied by multipliers

- Electromagnetic force and torque
- Stored energy
- Lorentz force by material
- Hysteresis loss by material
- Joule loss by material
- Flux linkage for a current condition
- Iron loss (iron loss analysis)
- Joule loss (iron loss analysis)
- Hysteresis loss (iron loss analysis)

MEMO The circuit conversion feature affects the output items for a circuit but does not affect the items listed above.

3-41. Two-Way Coupled Analysis

▶▶ Reference

- For more information, see "JMAG-Studio User's Manual," "Chapter 9. Setting Conditions of Magnetic Field Analysis," "Analysis Control: [Basic] Tab."
- For more information, see "JMAG-Designer User's Manual," "Chapter 29 Study Properties and Conditions of Magnetic Field Analysis," "Study Properties."
- For information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

3-41-1. Function

Specifies a two-way coupled analysis.

■ Related items

- Material (temperature dependent material, stress dependent material)
- Stress distribution condition
- Temperature distribution condition
- Displacement condition
- Partial model condition

■ Setting

- Two-way coupled analysis flag
- Remesh only air region

3-41-2. Important Notes Regarding the Use of This Function

- [Only remesh air region] can be selected when the displacement condition is set. Select this setting when the geometry of this model is large.

▶▶ Reference

- For more information about materials with magnetic property that are temperature dependent, see P.20 "2-3. Magnetizing Properties of a Soft Magnetic Material".
- For more information about materials with electric properties that are temperature dependent, see P.33 "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)".
- For more information about materials with electric properties that are stress dependent, see P.29 "2-3-5. Specifying the Magnetic Properties of a Stress Dependency".

3-42. Circuit Conversion

▶▶ Reference

- For more information, see "Analysis Control: Circuit Conversion" in Chapter 9 Setting Conditions of Magnetic Field Analysis in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 29 "Study Properties and Conditions of the Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-42-1. Function

The circuit conversion feature is used to set the values of a full model for the circuit of a partial FEM model, and the results of a full model are obtained after the calculation. With this feature, the values of a partial model do not need to be converted to the values of a full model because the circuit of a partial model can be treated as that of a full model. The input/output values for the circuit parameters are automatically converted based on the number of series/parallel connections.

■ Related items

- External circuit

■ Settings

- Conversion
  - Convert (periodic boundary)
  - Convert (series/parallel)
  - Do not convert
- Conversion factor other than periodic boundary
- Series/Parallel
- Convert (Series/Parallel)

■ Available modules

- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

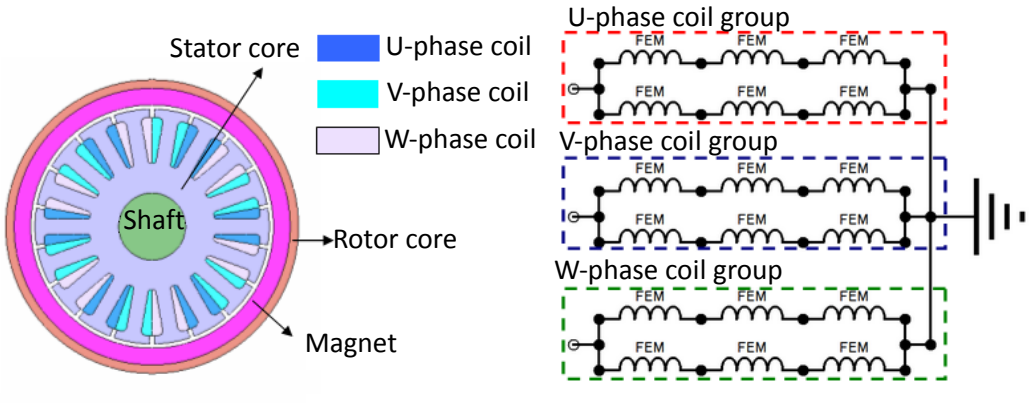
3-42-2. Important Notes Regarding the Use of This Function

- Determine the number of series/parallel connections based on how the circuit is configured in the full model.
- If selecting [Convert (Periodic Boundary)], specify the configuration of the partial model to be converted.
  - Series:
    Connects the partial models in series with the number of connections that is equal to the product of the values set for [Periodic Boundary] and [Other than Periodic Boundary].
  - Parallel:
    Connects the partial models in parallel with the number of connections that is equal to the product of the values set for [Periodic Boundary] and [Other than Periodic Boundary].
  - Antiperiodic Parallel:
    Connects the partial models sharing a antiperiodic boundary in series and then connects the sets of those series connections in parallel. The number of parallel connections is the same as determined by dividing the value set for [Periodic Boundary] by two, and the number of series connections, by multiplying the value set for [Other than Periodic Boundary] by two.
- The circuit conversion cannot be applied to the following parameter.
  - Number of turns for FEM coil components.
- The conversion factor for circuit is applied to the contact resistance for a brush/commutator component.
- The conversion factor for a circuit cannot be used when a cage component is set.
- Specify each circuit component by manually converging resistance to a section of coil for a partial model that is modeled or the voltage value to a section of a coil when the circuit conversion function is not used.

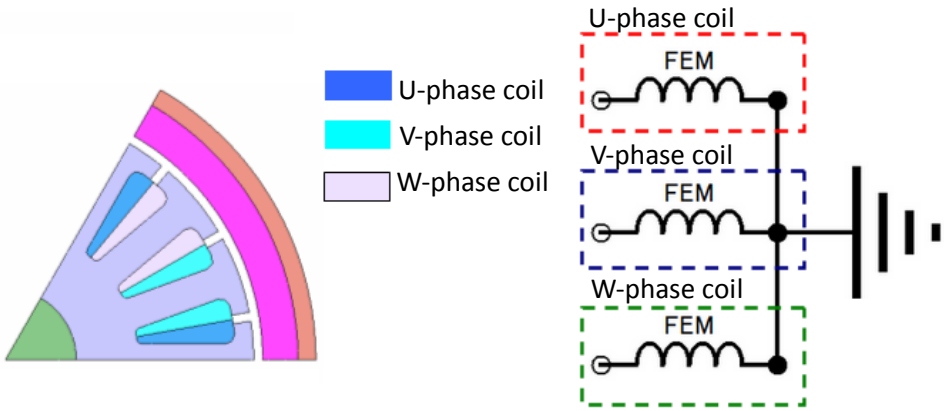
■ Example 1

- Shown below are the full model and the partial model of a motor. Due to the symmetry of the model, a 1/6 model can be created for this analysis.

  The FEM coil set for the partial model is configured in the full model to have three connections in series and two connections in parallel. In this case, you should select Convert (Series/Parallel), and then set Series Connection to "3" and Parallel Connection to "2" to represent the circuit converted to the full model.



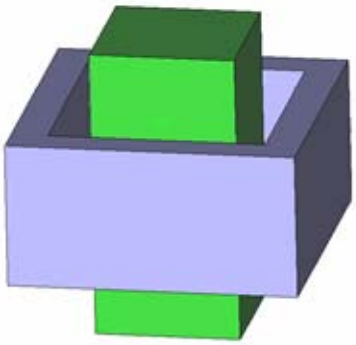
(A) Full model



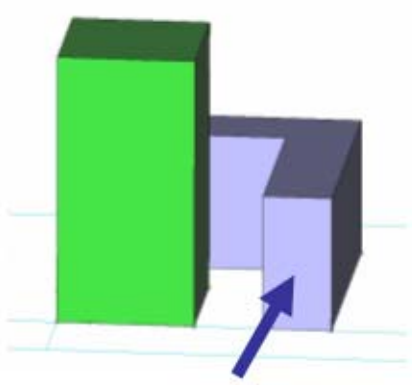
(B) Partial Model

■ Example 2

- Shown below are the full model and partial model of an electromagnet. Due to the symmetry of the model, a 1/8 model can be created for the analysis. In the partial model, the coil is halved horizontally and the top half which is modeled and bottom half are connected in series. Therefore, you should select Convert (Series/Parallel) as the conversion factor, and then set Series Connection to “8” and Parallel Connection to “1”. For the FEM coil component resistance, enter the value for the full model. For the number of turns, enter 1/2 of the value for the full model as it is set for the section represented in the partial model.



(A) Full model



(B) Partial model

3-42-2-1. Parameter and the conversion of calculated results

This section describes calculated results when [Convert (Synchronize with Periodic Boundary)] is selected as the circuit conversion. The following table is information on each item.

- Input
  Input value for circuit parameter and relations of value converted for one nth of a model
- Output
  The relation between the value obtained from an analysis of one nth of a model and results for a full model after conversion
- Argument of usrcir.f
  The relation between the value obtained from an analysis of one nth of a model and arguments for user subroutine (usrcir)

The conversion factor for other than periodic boundary is one.

Items			Series	Parallel	Parallel and Anti-Periodic
Input	FEM coil	Resistance value	R/n	R*n	(n/4) *R
	Resistor	Resistance value	R/n	R*n	(n/4) *R
	Capacitor	Capacitance value	C*n	C/n	(4/n) *C
	Coil	Inductance value	L/n	L*n	(n/4) *L
	Diode	Upper and lower limits of impedance	Rmax/n, Rmin/n	Rmax*n, Rmin*n	(n/4) *Rmax, (n/4) *Rmin
		Function	Coefficient K of the logarithmic function will be converted. K/n	Saturation current Is will be converted. Is/n	Both coefficient K of the logarithmic function and saturation current will be converted. K/2, (2/n)*Is
		Point sequence	The voltage will be converted. V/n	The current will be converted. I/n	The voltage and current will be converted. V/2, (2/n) *I

Items			Series	Parallel	Parallel and Anti-Periodic
Input	Voltage dependent resistor		The voltage and resistance value to be referenced will be converted. V/n, R/n	Only the resistance value will be converted. R*n	The voltage and resistance value to be referenced will be converted. V/2, (n/4) *R
	Voltage dependent switch		The voltage to be referenced will be converted. V/n	-	The voltage to be referenced will be converted. V/2
	Current dependent resistor		Only the resistance value will be converted. R/n	The current and resistance value to be referenced will be converted. R*n, I/n	The current and resistance value to be referenced will be converted. (n/4)* R, (2/n)* I
	Current dependent switch		-	The current to be referenced will be converted. I/n	The current to be referenced will be converted. (2/n)* I
	Supply voltage	Offset	V/n	-	V/2
		Point sequence	V/n	-	V/2
		Sin	Vamp/n	-	Vamp/2
		Exp	Vinit/n, Vamp/n	-	Vinit/2, Vamp/2
		Const	V/n	-	V/2
		Pulse	Vamp/n	-	Vamp/2
		Rectangle wave	Vamp/n	-	Vamp/2

Items			Series	Parallel	Parallel and Anti-Periodic
Input	Supply current	Offset	-	I/n	(2/n)* I
		Point sequence	-	I/n	(2/n)* I
		Sin	-	Iamp/n	(2/n)* Iamp
		Exp	-	Iinit/n, Iamp/n	(2/n)*Iinit, (2/n)* Iamp
		Const	-	I/n	(2/n)* I
		Pulse	-	Iamp/n	(2/n)* Iamp
		Rectangle wave	-	Iamp/n	(2/n)* Iamp
Out-put	Electric potential [V]		V*n	-	2*V
	Current [A]		-	I*n	(n/2) *I
	Flux linkage of FEM coil [Wb]		Φ*n	Φ*n	Φ*n
	Current of FEM coil [A]		-	I*n	(n/2) *I
Argument of usrcir.f			The voltage will be converted. V*n	The current will be converted. I*n	The voltage and current will be converted. 2*V, (n/2) *I

3-43. Power Source Component Control (JMAG-Designer)

▶▶ Reference

For more information, see “JMAG-Designer User's Manual,” “Chapter 29 Study Properties and Conditions of Magnetic Field Analysis,” “Study Properties.”

3-43-1. Function

Repeated computation is run on each step to determine voltage and electric current in a circuit, when using a power source component(172 page) in a circuit for a frequency response analysis.

The power source component control specifies settings for the calculation.

■ Related items

- Circuit(147 page): Power source component
- Coupled analysis (magnetic field and thermal)

■ Setting

- Tolerance
- Initial current
- Max count

3-43-2. Important Notes Regarding This Function

- The following setting is used for an analysis using the power source component.
  - Tolerance
    The acceptable range for the electrical power set in the power source component.
  - Maximum voltage, Maximum current
    The maximum voltage and the maximum current are specified in the power source component. The upper limit for the voltage and electric current applied to a circuit.
  - Max count
    The number of times a repeated computation is terminated. The calculation is terminated when the value set for repeated computation has exceeded, regardless of the settings specified for electrical power or acceptable range.
  - Initial current
    The initial current value for repeated computation.

3-44. Parallel Computing: Shared Memory Multiprocessing (SMP)

▶▶ Reference

- For more information, see "Analysis Control: Basic Settings" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 29 "Study Properties and Conditions of the Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-44-1. Function

The shared memory multiprocessor (SMP) is used to perform iterative calculations on multiple CPUs (cores) using parallel computing in a magnetic field analysis.

3-44-2. Important Notes Regarding the Use of This Function

The following should be noted when using the SMP.

- The parallel computing function (SMP) cannot be used when the direct method (FEM+BEM) is selected as the calculation method.
- The degree of parallelism can be selected from 2, 4, and 8.

3-44-2-1. License Type

The following two license types are available.

■ New license

The new license type requires one of the solver licenses (TR, DP, ST, and FQ) and the SMP licenses as many as the degree of parallelism.

For example, the number of licenses required for parallel computing in a 3D transient response analysis varies as follows:

- Two parallel processing: 1 TR solver license and 1 SMP license
- Four parallel processing: 1 TR solver license and 2 SMP licenses
- Eight parallel processing: 1 TR solver license and 4 SMP licenses.

■ Old license

The old license type requires the solver licenses (TR, DP, ST, and FQ) as many as the degree of parallelism.

For example, the number of licenses required for parallel computing in a 3D transient response analysis varies as follows:

- Two parallel processing: 2 TR solver licenses
- Four parallel processing: 4 TR solver licenses
- Eight parallel processing: 8 TR solver licenses.

When using an old license, specify the degree of parallelism for the environment variable "JMAG_OMP_SOLVER." If a number larger than the number of licenses is specified, the parallel calculation will terminate with an error. When specifying the environment variable, confirm the number of licenses that you own in advance.

MEMO The solver licenses for each analysis are as follows:
TR ---- Transient response analysis (3D)
DP ---- Transient response analysis (2D, axisymmetric)
ST ---- Static analysis (3D, 2D, axisymmetric)
FQ ---- Frequency response analysis (3D, 2D, axisymmetric)

MEMO The environment variable can be set as follows for using the parallel computing with an old license.

For Windows XP:

1. Select [Start] > [Control Panel].
   The [Control Panel] dialog box is displayed.
2. Double-click [System].
   The [System Properties] dialog box is displayed.
3. Click the [Advanced] tab, and then click [Environment Variables].
   The [Environment Variables] dialog box displayed.
4. Click [New] below the [User Variables] list.
   The [New User Variable] dialog box is displayed.
5. In the [Variable name] box, enter "JMAG_OMP_SOLVER".
6. In the [Variable value] box, enter the number of parallel computing used.
7. Click [OK].
8. In the [Environment Variables] dialog box, click [OK].
9. In the [System Properties] dialog box, click [OK].
10. Close the [Control Panel] dialog box.

For Linux:

1. Run the following commands.
   - C shell
     setenv JMAG_OMP_SOLVER The degree of parallelism
     * Separate the variable and value with a space.
   - bash
     export JMAG_OMP_SOLVER=The degree of parallelism
     * Connect the variable and value with an equal sign.

MEMO Please contact your local distributor for more information about recommended hardware environment for running parallel computing.

MEMO The following environmental variable needs to be specified for the AMD Opteron processor 8220 and Intel Xeon processor 5500 series
Variable name: SMP_FIRST_TOUCH
Value: 1

3-45. Parallel Computing: Distributed Memory Multiprocessor (DMP)

▶▶ Reference

- For more information, see “Analysis Control: Basic Settings” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-45-1. Function

The distributed memory multiprocessor (DMP) is used to perform iterative calculations on multiple machines on the same network using parallel computing in a magnetic field analysis.

3-45-2. Important Notes Regarding the Use of This Function

The following should be noted when using the DMP.

- The DMP complies with MPI (Message Passing Interface). The following MPI is supported (as of December 2009).
  - MPICH2 (1.0.8 or later)
- The parallel computing function (DMP) cannot be used when the direct method (FEM+BEM) is selected as the calculation method.
- This setting cannot be used with the time periodic explicit error correction function.
- The degree of parallelism that can be selected are 2,4, and 8.
- A solver license (TR, DP, ST, FQ) and a DMP license specified for a degree of parallelism is required.
  For example, the following licenses are required for parallel computing in a 3D transient response analysis.
  - Two parallel processing
    One “TR” solver license and one DMP license
  - Four parallel processing
    One “TR” solver license and two DMP licenses
  - Eight parallel processing
    One “TR” solver license and four DMP licenses

Important **Parallel computing for distributed memory multiprocessing (DMP) and parallel computing for shared memory multiprocessing (SMP) can use the same license.**

- The same version of solver needs to be installed in all of the machines that are used for the calculation. The installation directory of solver needs to be the same.
- The network environment where all of the machines are connected through 1GB Ethernet is required. If you use other network environment, please contact your distributor.
- If a network is used for other purposes than the DMP (for example, sending/receiving e-mails, internet, and parallel computing other than JMAG etc.) the performance of parallel processing slows down.
- The same UID/PWD are required for each machine that is used for the DMP parallel computing.
  - If all of the machines use the same domain, a domain user can be used.
  - If not, the local user of the same UID/PWD is required for all of the machines.
- The operation system of the machines that are used as calculation node must be either Windows or Linux. The parallel computing cannot run in the environment where Windows and Linux are mixed.
- If JMAG-Remote System is used with the DMP parallel computing, the management server needs to be specified using the aforementioned user for JMAG-Remote System, and JMAG-Remote System needs to be restarted. The procedure is as follows.

■ **For Windows:**

i. **Select [Start] > [Control Panel].**

The [Control Panel] dialog box is displayed.

ii. **Double-click [Administrative Tools].**

If the [Control Panel] dialog box is displayed in the category view, click [Performance and Maintenance], and then click [Administrative Tools].

iii. **Double-click [Services].**

A list of services is displayed.

iv. **Double-click [JMAG-Remote System].**

The [JMAG-Remote System Properties] dialog box is displayed.

v. **Click the [Log On] tab.**

vi. **Select [This account].**

vii. **Enter the user information in the [This account] box and [Password] box.**

viii. **Click [OK].**

■ **For Linux:**

Start JMAG-Remote System (execute JMAGRemoteSystemStart.sh) with the user that is used to start mpd of MPICH.

3-46. Use GPU (JMAG-Designer)

▶▶ Reference

For more information, see "JMAG-Designer User's Manual," "Chapter 29 Study Properties and Conditions of Magnetic Field Analysis," "Study Properties."

3-46-1. Function

The calculation time can be shortened by using the GPU (Graphics Processing Unit) that is installed onto the machine to process the calculation in JMAG.

This function is useful when handling an analysis model of large size.

3-46-2. Important Notes Regarding the Use of This Function

- The GPU can be used in Windows 64bit modules.
- The GPU can be used in a static analysis or a transient response analysis.
- The GPU function can also be used for coupled analysis. However, only the calculation a magnetic field analysis can be shortened by using this function.
- The GPU cannot be used when the direct method (FEM+BEM) is selected as the calculation method.
- The GPU function supports double precision calculation and tesla/Fermi architecture supported by NVIDIA GPU. The calculation ends in an error when the GPU function is used on a machine that does not have the supported GPU installed.

MEMO Information about the GPU that is supported with JMAG is described in the JMAG Web site. Please visit us on the Web at:

`http://jmag-international.com`

3-47. Steady-state Approximate Transient Analysis

▶▶ Reference

- For more information, see “Analysis Control: Basic Settings” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-47-1. Function

In a steady-state approximate transient analysis, the calculation at the first step is run in an almost steady state, and a normal transient response analysis (TR, DP) is run from the second and later steps based on the results obtained from the first step using the restart feature. This feature allows an analysis to settle into a steady state relatively quickly.

■ Settings

- Type of analysis model
- Slip

■ Available modules

- Transient response analysis (TR, DP)

3-47-2. Important Notes Regarding the Use of This Function

The following should be noted in a steady state approximate transient analysis.

- This feature is suitable for an analysis that has a long-lasting transient state such as an analysis of an induction motor.
- Two or more steps need to be specified.
- The [X-axis] type and [Function] type need to be set to [Time] and [Sin] for the current condition, electric potential source component, current source component, three-phase current source component, and three-phase electric potential source component. The same frequency needs be specified for all of the above.
- Magnet cannot be used.
- In an analysis of an induction motor, the slip needs to by specified. The slip should be a real number greater than zero and smaller than or equal to 1. For example, if a 60 Hz current source is used and the slip is set to 0.1, the slip frequency is 6 Hz.
- The [Output 1st Step as Steady Result] can be used when running an induction motor analysis.
  - Outputs the steady state result obtained from a steady-state approximate transient analysis that is run in the first step when it is on.
  - Outputs the transient state result obtained from a steady-state approximate transient analysis that is run in the first step when it is off.

3-48. Time Periodic Explicit Error Correction (JMAG-Designer)

▶▶ Reference

For more information, see “JMAG-Designer User's Manual,” “Chapter 29 Study Properties and Conditions of Magnetic Field Analysis,” “Study Properties.”

3-48-1. Function

This function reduces the time to reach steady state by removing errors in an answer for each 1-period or half period that is specified.

It is useful for analyzing an induction motor, synchronous motor, and reactor with electric potential source etc.

■ Related items

- Step control

■ Settings

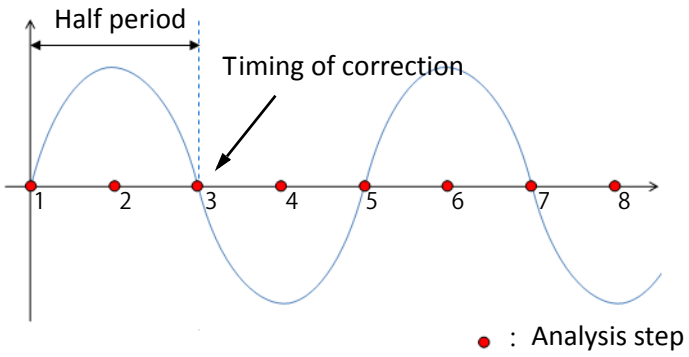
- Anti-Periodic or Periodic
- Use Reverse Correction Type for the Rotor
- Use relaxation factor
- Correction Frequency or Correction Interval
- Max. Corrections

■ Modules that can be used with this function

- Transient response analysis (TR, DP)

3-48-2. Important Notes Regarding the Use of This Function

- This function is less effective for an analysis using a current condition or current source component as a power supply.
- This function cannot be used for the following.
  - Selecting [FEM+BEM] for calculation method
  - An external circuit link condition is used
  - Parallel computing (Distributed Memory Multiprocessing (DMP))
  - Using a function to generate mesh (patch mesh) during analysis
- Always select [Anti-periodic] as the correction type if the analysis is antiperiodic. This function is less effective when [Periodic] is selected.
- Adjust the step control so that the timing that is corrected (1-period or half period of power supply) matches with a time of an analysis step that exists when using this function.



Example: Waveform and analysis step of power supply
The timing that is corrected (half period) matches with the timing in step 3.

- The results with steady state needs to be set when using results that used this function to run iron loss analysis or one way coupled analysis.

3-49. Displacement Current (JMAG-Studio)

3-49-1. Function

The displacement current can be taken into account in frequency response analysis (FQ). The relative permittivity needs to be specified. The electric field (V/m) can be obtained as a result.

3-49-2. Important Notes Regarding the Use of This Function

- If an FEM conductor condition is set, the displacement current cannot be specified.

▶▶ Reference

For more information, see “Analysis Control: Basic Settings” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.

3-50. ICCG

▶▶ Reference

- For more information, see "Analysis Control: Setting of ICCG Method" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 29 "Study Properties and Conditions of the Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-50-1. Function

Specifies the tolerance for aborting calculation and the maximum number of iterative calculations when running iterative calculation with the ICCG method. To modify a convergence tolerance at each step, use "Step control."

■ Related items

- None

■ Settings

- Acceleration coefficient
- Convergence tolerance
  - Set automatically at every stage
  - Set manually at every
  - Constant
- Divergence tolerance
- Maximum number of iterations
- ICCG control
  - Use the value at minimal residue
  - Restart
- Calculation method

■ Available modules

- All of the modules of magnetic field analysis

■ Output items

- None

3-50-2. Important Notes Regarding the Use of This Function

3-50-2-1. Acceleration Coefficient

Change when iteration does not converge. The range 1.02 to 1.5 is recommended. The default value is 1.05. When [Automatic] is selected, JMAG searches for the best result and uses it

3-50-2-2. Convergence Tolerance

Set the convergence tolerance for the ICCG method, which is run at each iteration of the nonlinear iterative calculation. It is operative only when the ICCG method is used during the execution of nonlinear calculations.

There are three types of setting methods as follows.

■ Automatic setting of the ICCG method convergence tolerance at each iteration of the nonlinear iterative calculation

An appropriate ICCG method convergence tolerance is automatically determined for each iteration of the nonlinear iterative calculation, within the designated range of values given by the set maximum and minimum values. Calculations can be speeded up compared to when [Constant] is selected. The default values are the recommended values (maximum value: 1e-006, minimum value: 1e-008).

The maximum value is used for the ICCG method convergence tolerance during the first iteration of the nonlinear calculation, with the ICCG method convergence tolerance getting more stringent as the calculation continues.

In rare cases, the final ICCG method convergence tolerance does not get sufficiently small.

In such cases, reduce the value in the [Maximum Value] text box.

■ Manual setting of the ICCG method convergence tolerance at each iteration of the nonlinear iterative calculation (old: step control function of the ICCG method pane)

An appropriate ICCG method convergence tolerance for each iteration of the nonlinear iterative calculation is set in step unit.

If this method is selected, perform the following settings.

i. Set the iteration number of the nonlinear iterative calculation and its appropriate ICCG method convergence tolerance, and make a convergence tolerance table.

Example: Convergence tolerance table.

	1; Number of Nonlinear Iteration	2; ICCG Convergence Tolerance
1st Point	2	1.00e-06
2nd Point	4	1.00e-07

For the example above, refer to the table for each nonlinear iteration as follows.

- Nonlinear iterative calculation: the 1st

  Since the first iteration is undefined in the table, the value set in [Constant] is used in the ICCG method iterative calculation for the ICCG method convergence tolerance.
- Nonlinear iterative calculation: the 2nd - 3rd

  1e-6 is used as the convergence tolerance in the ICCG method iterative calculation.
- Nonlinear iterative calculation: the 4th and later

  1e-7 is used as the convergence tolerance in the ICCG method iterative calculation.

The iteration numbers of nonlinear calculation in the convergence tolerance table may be discontinuous numbers as long as they are in an ascending order.

ii. Set the step and the convergence tolerance table to use for that step, and make a step control table.

Example: Step control table

	1; Step	2; Convergence Tolerance Table
1st Point	3	1
2nd Point	5	2

For the example above, refer to the table for each step as follows.

- The 1st -2nd step

  Since the number of the convergence tolerance table is undefined, the convergence tolerance set in [Constant] is used in the ICCG method iterative calculation for all the nonlinear iterative calculation.
- The 3rd - 4th step

  Refer to convergence tolerance table 1.
- The 5th step -

  Refer to convergence tolerance table 2.

The steps in the step control table may be discontinuous numbers as long as they are in an ascending order.

■ Using a constant value at all times for the ICCG method convergence tolerance

Apply the set ICCG method convergence tolerance for all the iterations in the nonlinear iterative calculation.

3-50-2-3. Divergence Tolerance

If the tolerance exceeds the calculation abort tolerance, the iterative calculation is considered to be divergent, and calculation is aborted. The default value is 1e+20.

3-50-2-4. Maximum Number of Iterations

Change when convergence is not achieved with the default value. The default is 5000 cycles.

3-50-2-5. ICCG Control

■ Use the optimum value when not converged

Specify whether or not to use the value at minimum tolerance as the solution when the iterative calculation has not converged with the ICCG method.

- If [Use the value at minimal residue] is selected, value at minimum tolerance is used as the solution.
- If [Use the value at minimal residue] is cleared, the value at the last iterative calculation is used as the solution.

■ Restart

If the iterative calculation with ICCG method does not converge, the iterative calculation is repeated using the value in the last step as the initial value. (This is not now supported.)

■ Calculation method

Specify whether to add the scalar potential as the unknown to a conductor. If the model consists of mostly conductors, the convergence of the calculation can be faster.

Select a calculation method from the following.

- A method:
  Do not add the scalar potential to a conductor. The default setting is A method. If the model does not have any conductors, A method is used.
- A-phi method 1:
  Add the scalar potential as the unknown in the conductor for the analysis taking into account the eddy currents. This method is used in the version before Studio version 9.1 and Designer version 5.0. If the model consists of mostly conductors, the convergence of the calculation can be faster.
- A-phi method 2:
  In most cases, the efficiency of the calculation is improved.

3-51. Nonlinear Calculation

▶▶ Reference

- For more information, see “Analysis Control: Setting of Nonlinear Calculating Method” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-51-1. Function

Sets parameters of nonlinear calculation.

■ Related items

- None

■ Settings

- Number of nonlinear iteration
- Nonlinear iterative convergence tolerance
- Calculation method of nonlinear iteration
- Flag to use the strict judgment against the convergence determination of nonlinear iterative calculation
- Flag to use the speed-up analysis scheme for nonlinear iterative calculation
- Flag to use Correct BH Curve for Frequency Response Analysis

3-51-2. Important Notes Regarding the Use of This Function

3-51-2-1. Maximum Nonlinear Iterations

Sets the maximum iteration number of nonlinear calculation. A nonlinear calculation will be iteratively carried out until the solution would converge.

When the relative variable of a magnetic field before and after the iteration reached a specified iterative tolerance, the convergence calculation is terminated. Or, when the nonlinear iterations reached a specified nonlinear maximum number of iterations, the convergence calculation is terminated.

- The maximum iteration number of nonlinear calculation must be set for analyses that deal with nonlinear physical properties.
- The default value is 15.

3-51-2-2. Convergence Tolerance

Sets the nonlinear calculation convergence tolerance.

- This is the value at which the nonlinear calculation is determined to have converged. If the variation of the solution is below the convergence tolerance, it is determined to have converged.
- The default value is 1e-3.

3-51-2-3. Calculation Method for Nonlinear Iteration

Selects the method of nonlinear iterative calculation.

- Either the Newton-Raphson iterative method or Successive Iterations method can be selected.

■ Newton-Raphson method

Normally select this method.

When this method is selected, set the convergence tolerance of nonlinear iteration to around 1e-3 and the maximum number of iterations to about 15 times.

When the Newton-Raphson method is selected, relaxation factor is available. When nonlinear material is used for the magnetizing properties and the tilt of the B-H curve (relative permeability) does not monotonously decrease, convergence may be difficult. In such cases, using the relaxation factor can improve convergence.

■ Relaxation Factor 2

This is the default (recommended) value for relaxation factor. Select this when using an B-H curve of S-shape (that has one inflection point) such as material database.
Note that if a diode component is used for an external circuit and the default value is selected, convergence may be difficult. When using a diode component, select [Not Used] or [Relaxation Factor 1].

■ Not Used

When the tilt of the B-H curve (relative permeability) is using only the magnetizing properties of monotonous decrease, the relaxation factor is not required. Using the relaxation factor may increase the number of nonlinear iterative calculations.
Select [Not Used] if convergence is difficult due to use of a diode component in the circuit.

■ Relaxation Factor 1

This is a counterpart method between [Relaxation Factor 2] and [Not Used]. Try [Relaxation Factor 1] if both [Relaxation Factor 2] and [Not Used] cannot make the calculation converge.

■ Successive method

When performing the calculation on the superconductor, select the successive iteration method.

When this method is selected, set the convergence tolerance of nonlinear iteration to around 1e-5 and the maximum number of iterations to about 100 times.

MEMO Calculating superconductors is only supported in JMAG-Studio.
For more information about superconductors, see "2-10. Superconductivity Properties (JMAG-Studio)" on page 64.

3-51-2-4. Using the Strict Criteria for Convergence Tolerance

In typical nonlinear iterations, a calculation terminates and outputs the analytical results at the time when the magnetomotive force became small to a certain extent. In an analysis required to obtain fine eddy current distribution, however, the case may occur where a usual judgment is insufficient.

In that case, if this check box is selected, a judgment whether non-magnetic iterations should be terminated can be strictly made only by testing the fluctuation of the fields without using the magnitude of electromotive force.

3-51-2-5. Using the High Speed Solver

When using non-linear material for the transient response analysis (2D, 3D, and axial symmetric), this function can shorten the nonlinear calculation time.

The speed-up analysis scheme can apply to all analysis such as a single analysis (transient response magnetic field analysis only), coupled analysis, the analysis linking with the external program.

Note the following.

- In the case of the analysis with motion, this function is not recommended for analysis if displacement per step is large. This may decrease the calculation speed.
- When using the patch mesh function (using both motion condition and [Generate Mesh While Running Analysis]), this function is not available.
- This function is not recommended for the no-load analysis without the current condition and the current/electric potential source in the circuit. The calculation result of cogging torque and induced voltage may not be accurate.

3-51-2-6. Correcting the B-H Curve for Frequency Response Analysis

- In the frequency response analysis, either of the following two methods can be used for the nonlinear magnetization curve.

■ Do not correct the nonlinear magnetization curve

The amount of magnetic field and the amount of magnetic flux density can be related using the nonlinear magnetization curve. The Joule loss cannot be obtained accurately because magnetic saturation is not taken into account.

■ Correct the nonlinear magnetization curve

Even if the magnetic field changes sinusoidally, the magnetic flux density does not change sinusoidally due to magnetic saturation. The effect of magnetic saturation needs to be taken into account. Correct the nonlinear magnetization curve by taking into account magnetic saturation. This method is used to obtain the accurate Joule loss.

To use the later method, [Use Correct BH Curve for Frequency Response Analysis] needs to be enabled.

3-52. Post Output

▶▶ Reference

- For more information, see "Analysis Control: Setting of Output" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 29 "Study Properties and Conditions of the Magnetic Field Analysis" in the JMAG-Designer User's Manual.

3-52-1. Function

The post output settings are used to select output items as analysis results.

The size of a plot file can be specified.

Output items can be selected for each material.

The analysis steps to export the results can be specified.

■ Related items

- Frozen permeability (JMAG-Designer)

■ Settings

- Output items
  - Magnetic flux density
  - Magnetization/magnetic field
  - Current density
  - Current (beam)
  - Demagnetizing ratio
  - Joule loss density
  - Surface force density
  - Lorentz force density
  - Electric field
  - Permeance coefficient
  - Displacement
  - Relative permeability *JMAG-Designer only
  - Differential permeability *JMAG-Designer only
  - Vector potential
  - Nodal force

  - Restart file (*.sav)
  - Stored energy
  - Stress
  - Initial magnetization for permanent magnet * JMAG-Designer only
  - Mark display indicating the convergence status of the solver
- Control of output intervals
- Output control of the results for each material * JMAG-Studio only
- Output control of the results for each part *JMAG-Designer only
- Control of the plot file size *JMAG-Studio only

■ Available modules

- All of the modules of magnetic field analysis

3-52-2. Important Notes Regarding the Use of This Function

3-52-2-1. Controlling Output Items

The following items can be exported.

- Stored energy: Stored energy is calculated using the following equation.

$$\text{energy} = \int \mathrm{H}\,\mathrm{dB}$$

Stored energy is used to obtain inductance. The following equation indicates the relation between stored energy and inductance (*U*: stored energy, *L*: inductance, current: *I*).

- Linear magnetizing properties: $\mathrm{U} = \frac{1}{2}\mathrm{L}\mathrm{I}^2$
- Nonlinear magnetizing properties: $\mathrm{L} = \frac{\Delta \mathrm{U}}{\mathrm{I}\Delta \mathrm{I}}$

- Magnetic flux density
- Magnetization/magnetic field
- Current density
- Current (beam)
- Demagnetizing ratio
- Joule loss density
- Surface force density

- Lorentz force density
  The Lorentz force density is outputted for the 0-order component, and 2-order component when a magnetic field analysis (frequency response analysis) is specified.
  - The 0-order component is used when running a structural analysis (static analysis) that is using a result from a magnetic field analysis (frequency response analysis).
    *One-way coupled analysis
  - The 0-order component and the 2-order component is used when running a structural analysis (frequency response analysis) that is using a result from a magnetic field analysis (frequency response analysis).
    *One-way coupled analysis
- Electric field
- Permeance coefficient
- Displacement
- Relative permeability *JMAG-Designer only
- Differential permeability *JMAG-Designer only
- Nodal force
  The nodal force is outputted for the 0-order component, and 2-order component when a magnetic field analysis (frequency response analysis) is specified.
  - The 0-order component is used when running a structural analysis (static analysis) that is using a result from a magnetic field analysis (frequency response analysis).
    *One-way coupled analysis
  - The 0-order component and the 2-order component are used when running a structural analysis (frequency response analysis) that is using a result from a magnetic field analysis (frequency response analysis).
    *One-way coupled analysis
- Vector potential * Transient response analysis (2D/axisymmetric) only
- Stress
- Initial magnetization for permanent magnet
  The magnetic flux density or magnetic field result of initial magnetization in M_0 direction can be confirmed, by using the user component function.

  ▶▶ Reference
  For more information about the user component function, see "JMAG-Designer User's Manual."
- Restart file (*.sav)
- Display/hide "* " and "· " that indicate the convergence status of the solver

  ▶▶ Reference
  - For more information about the output items, see P.410 "3-59. Output Items of Magnetic Field Analysis".

- For more information about structural analysis using results from a magnetic field analysis (frequency response analysis), see "10-6-2. Magnetic Field Analysis (Frequency) -> Structural Analysis (Static/Frequency)" on page 807.

3-52-2-2. Controlling Output Materials (JMAG-Studio) / Controlling Output Parts (JMAG-Designer)

In JMAG-Studio, the results of each material can be exported to a result (plot) file. The check box of each material needs to be selected to export the results of the material to a plot (or jplot) file.

In JMAG-Designer, the results of each part can be exported to a result (or jplot) file. The check box of each part needs to be selected to export the results of the part to a jplot file.

Important **If the check box of a material (part) is cleared, all of the element and nodal values belonging to the material (part) are not exported. The check box of a material (part) needs to be selected to view vector and contour distributions in the parts.**

3-52-2-3. Controlling the File Size (JMAG-Studio)

Specify the [Division of File (MB)] text box.

When results are output exceeding a set value, a PLOT file is automatically divided.

For example, when a calculation is made under the name “Data”, files named “Data.plot001”, “Data. plot002”, ... are created. Since the whole of these files compose a set of results of an analysis, do not delete any part of them.

3-52-2-4. Controlling Output Intervals

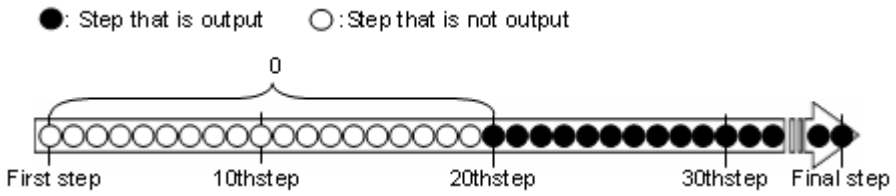
The size of a result file (*.jplot, *.plot) can be minimized by controlling the output to the file by step. There are three ways to control the result output.

■ All steps

Select this to output the results for all steps.

■ Specified steps

If you select this option, the results for all steps from the step you specify here will be output. For example, if you specify "20", the results for step 20 and all steps thereafter will be output as shown below.



■ Advanced setting (Specifying the output interval with point sequence)

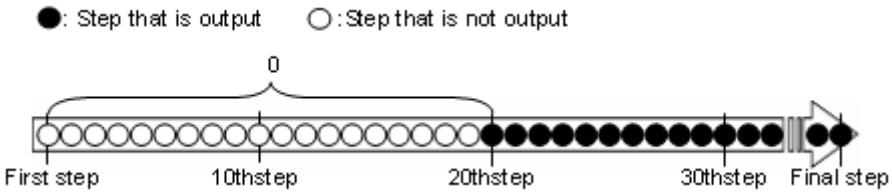
With this option, you can define the step intervals for output using a table.

- The output interval can be defined more specifically. For example, you can set it to output the results every 5 steps up to step 20, and from step 20 every 2 steps.
- If you set the output interval to "0", the steps before the specified step are not output.
- If you set the output interval to "1", all steps before the specified step are output.
- The step specified with a table and the steps thereafter are all output.
- Specify the table so that the step IDs are in ascending order.

Example)

■ 1) Output Interval Table A

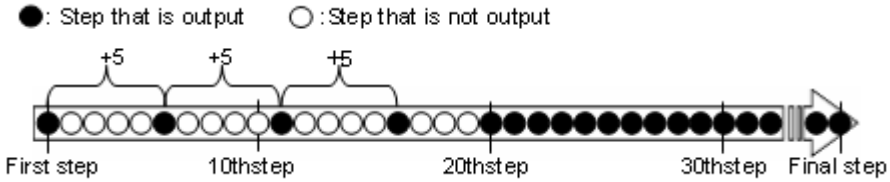
	1: Step to End Control	2: Output Interval
1st point	20	0



- From the first Step to the 19th Step, no output is made.
- From 20th Step to the final Step, all steps are output.

■ 2) Output Interval Table B

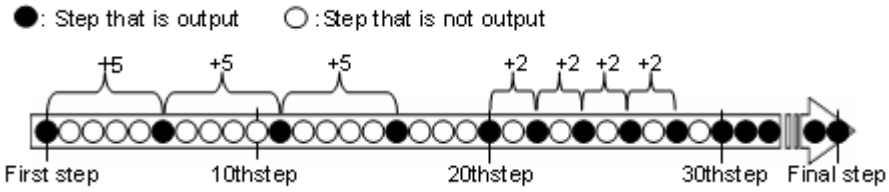
	1: Step to End Control	2: Output Interval
1st point	20	5



- From the first Step to the 19th Step, output is made at five-step intervals.
- From the 20th Step to the final Step, all steps are output.

■ 3) Output Interval Table C

	1: Step to End Control	2: Output Interval
1st point	20	5
2nd point	30	2



- From the first to the 19th Step, output is made at five-step intervals.
- From the 20th Step to the 29th Step, output is made at two-step intervals.
- From the 30th Step to the final step, all steps are output.

3-53. Controlling User Subroutine

▶▶ Reference

- For more information, see “Analysis Control: User Subroutine” in Chapter 9 Setting Conditions of Magnetic Field Analysis in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-53-1. Function

Selects the user subroutine to be used in the analysis.

User subroutine is a program that a user creates to express a function that JMAG cannot realize by itself and to perform a calculation. The user subroutine must be written in Fortran or C++.

■ Related items

- User subroutine

■ Settings

- Volume of arrays for the use of user (word)
- Flag for calculating magnetizing process
- User subroutine DLL to be used
- User subroutine to be used

■ Available modules

- All of the modules of magnetic field analysis

3-53-2. Important Notes Regarding the Use of This Function

- The array size available for the user subroutine *usrfma* can be set by the user array size (size of user array (usrfma) (word)). This array takes a real type of array that can be used by the user. Enter the array size in integer. The default value is 0. The number 1 corresponds to one real number.
- Specify the user subroutine DLL to be used in the analysis in the [Select User Subroutine DLL] group box of the [User Subroutine] pane.

  When using a user subroutine DLL on a remote machine for the analysis run on the remote system, the user subroutine DLL needs to be added to the list.
- The user subroutine DLL to be used in the analysis must be stored in the following locations:
  - Windows: JMAG Installation Directory\usrsub\magnet
  - Linux: /works/solver/mod/magnet/
- If you want to distribute the calculation that uses the user subroutine DLL, place the user subroutine DLL if using a Windows machine, or shared library if using a Linux machine, on all CPU nodes in the CPU group to be used for the distributed calculation.
- The user subroutine is enabled by selecting the subroutine check box in the [User Subroutine] group box of the [User Subroutine] pane.

  The user subroutine that does not have any check box on this pane becomes available under the other conditions, such as the occasion when the user-defined material was chosen in the materials property.

3-54. Snapshot Calculation

▶▶ Reference

- For more information, see “Analysis Control: Basic Settings” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see Chapter 29 “Study Properties and Conditions of Magnetic Field Analysis” in the JMAG-Designer User's Manual.

3-54-1. Function

The snapshot calculation is a feature that distributes the calculation to multiple machines by the step when the analysis contains separate processes for each step.

The separate processes means that the result for a step does not affect the result for the next step. This applies to the analyses where the current condition is used.

■ Settings

- Number of retries for failed analysis
- Retry Mode * JMAG-Studio only
  - Recalculates only failed analysis
  - Recalculates all the analysis
- Number divided by the number of steps

■ Available modules

- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

3-54-2. Important Notes Regarding the Use of This Function

- The snapshot calculation cannot be used for the analysis where the data of the previous step is required such as follows:
  - An analysis using eddy current (transient response analysis)
    Eddy currents are taken into account in the analysis when the [Allow Eddy Currents] check box is selected for the material property settings.
  - Analysis using a circuit
  - Analysis using a user subroutine or the motion equation of a motion condition.

- If the distributed calculation for a step does not complete successfully, the calculation will be automatically repeated as many times as specified for [Number of Re-trial] on another machine.
- The JMAG-Remote System needs to be installed to run the snapshot calculation.
- The snapshot calculation function cannot be used for two way coupled analysis.

3-54-3. Executing Analysis using Third-party Job Management Systems

The snapshot calculation has the following modes built-in.

- A mode to bundle and create the input files (*.jcf) used for analyses.
- A mode to remotely execute magnetic field analyses using input files (*.jcf).
- A mode for post-processing based on the magnetic field analysis results.

Analysis can be run and managed using a third-party job management system using these modes.

Important **Processes for forwarding input files that are generated and the result files obtained after an analysis as well as adding jobs into a job management system need to be defined by script created by the user.**

■ Call command

The following command is executed from the command line to run a snapshot calculation (module name: SnapShot) in a job management system.

```
~/SnapShot.exe
```

Furthermore, use the following arguments based on the mode.

- Mode to create the JCF file for the snapshot calculation

  ```
  Argument: "-induct_jcf"
  ```

- Mode to run the magnetic field analysis (remote execution)

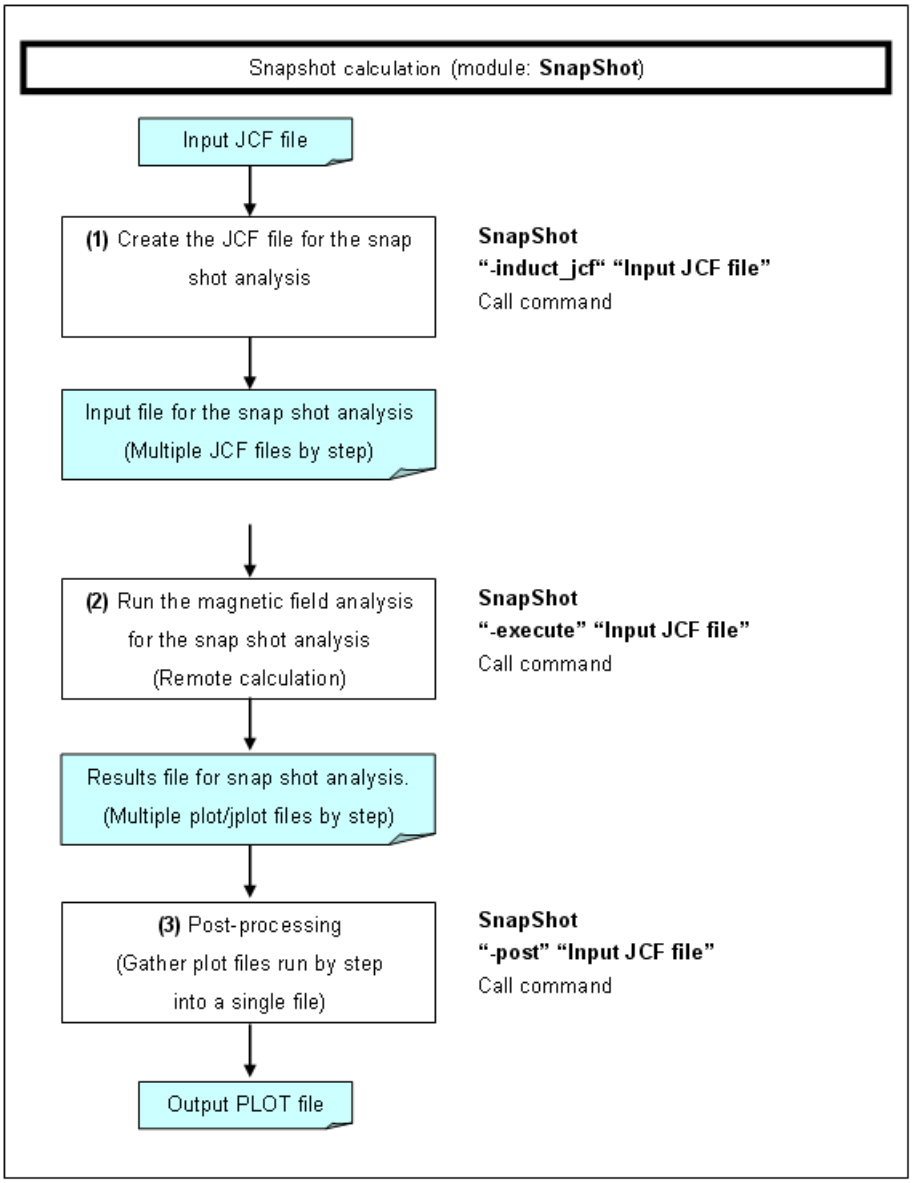
  ```
  Argument: "-execute"
  ```

- Post-processing mode (bundle the plot and jplot files run as snaps in a single file).

  ```
  Argument: "-post"
  ```

■ Call flow

The call flow for job management systems are as follows:


Snapshot calculation (module: SnapShot)
Input JCF file
(1) Create the JCF file for the snap shot analysis
SnapShot
"-induct_jcf" "Input JCF file"
Call command
Input file for the snap shot analysis
(Multiple JCF files by step)
(2) Run the magnetic field analysis for the snap shot analysis
(Remote calculation)
SnapShot
"-execute" "Input JCF file"
Call command
Results file for snap shot analysis.
(Multiple plot/jplot files by step)
(3) Post-processing
(Gather plot files run by step into a single file)
SnapShot
"-post" "Input JCF file"
Call command
Output PLOT file

3-55. Restart

▶▶ Reference

- For more information, see "Analysis Control: Basic Settings" in Chapter 9 "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Running the analysis using one study" in Chapter 40 "Running Analysis" in the JMAG-Designer User's Manual.

3-55-1. Function

An analysis can be performed using the results of previous analysis as initial values using the restart function.

This function is useful when a single analysis is to give a large plot file because the analysis can be divided into steps and run by steps.

This function is also useful when the steady state has not been reached after a transient response analysis (TR, DP) because the analysis can be continued to run from the point where the analysis ended.

With this function, the calculation time can be reduced by running a frequency response analysis to obtain values approximate to the steady state, and then transferring the results to a transient response analysis where it is difficult to reach the steady state.

■ Related items

- Step/Frequency

■ Settings

- Types of restart
  - From the transient response analysis
  - From the frequency response analysis

■ Available modules

- Transient response analysis (TR, DP)
- Frequency response analysis (FQ)

■ Output items

- None

3-55-2. Types of Restart

There are two types of restart.

■ TR (DP) => TR (DP)

A transient response analysis (TR, DP) can be continued to run by using the results of previous analysis.

■ FQ=> TR (DP)

A transient response analysis (TR, DP) can be continued to run by using the results of a frequency response analysis where the steady state is calculated. This function is useful for machines such as induction machines where the transient state continues for a long period of time.

The restart function cannot be used for analyses with different dimensions. For instance, a 2D transient response analysis (DP) cannot be run using the results of a 3D frequency response analysis (FQ).

MEMO In the case of "FQ => TR(DP) Restart", the phase of a sinusoidal wave in a frequency response analysis needs to be set to θ_1- 90 degrees relative to the phase θ_1 of a sinusoidal wave of magnetomotive force in a transient response analysis.

3-56. User Subroutine

Some of the subroutines in JMAG so-called user subroutines are open to users, and these can be customized.

▶▶ Reference

- For more information, see “Analysis Control: User Subroutine” in Chapter 9 “Setting Conditions of Magnetic Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 29 “Study Properties and Conditions of the Magnetic Field Analysis” in the JMAG-Designer User's Manual.
- In the case of magnetic field analysis, see "3-53. Controlling User Subroutine" on page 323.
- In the case of thermal analysis, see "4-27. Controlling User Subroutine" on page 468.
- In the case of iron loss analysis, see "12-3-9. Settings in the [User Subroutine] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)" on page 902.

3-56-1. Types of User Subroutine

■ ***magusr***

Magnetizing properties can be defined.

■ ***conusr***

Allows for defining electric properties such as conductor aeolotropies. This is also used to give aeolotropies to electrical conductivity of superconductors To make this subroutine available, select user subroutine for material electric properties.

■ ***usrfma***

Allows for describing equations of motion to define displacement and time when the object to be analyzed changes its location with each motion or analysis step. To make this subroutine available, select user subroutine for motion condition.

■ *usrfm2*

When the equation of motion is used in a motion condition, set the mass, attenuation constant, spring constant, etc., so that they can be modified during a calculation. To make this subroutine available, select the check box in the user subroutine of analysis control.

■ *usrcir*

Use this subroutine when required to set an arbitrary resistance, capacitance, inductance, or ON/OFF in the calculation of an external circuit. In case user subroutine component is used in the circuit, this subroutine can be used.

■ *usrstp*

Allows for forcing to change step value or time in analysis process.

■ *usrfle*

Allows for opening any file. This is called only at the analysis start and end. This should be used when creating a new file and writing it with other subroutines.

■ *usrxv*

Allows for setting arbitrary values as point sequence X-axis values. This function is called when determining Y-axis values from X-axis values in the current point sequence, current density point sequence, potential point sequence, and circuit component point sequence.

■ *usrlos*

Output value of Joule loss can be corrected for each element.

■ *usrmtx*

Coordinate System to be used in the Anisotropic Material can be arbitrarily set. To make this subroutine available, select the definition in user subroutine when specifying anisotropy/lamination direction for material magnetizing properties.

■ *hsusr1*

This subroutine generates a file that contains the information of each element. The file can be referred by the other subroutine for magnetizing property and electrical property.

■ *hsusr3*

The values of user-defined arrays can be changed on completion of calculations at each step or when the plot file is exported.

■ *hsusr4*

The correction coefficient of flux density, bfac, can be defined.

■ *usrhys*

Allows the hysteresis loss density to be defined.

■ *usrene*

With magnetizing properties defined by *magusr*.f, the energy density must be defined with this subroutine to calculate the electromagnetic force and accumulated energy. To make this subroutine available, select user Subroutine for magnetizing properties of material.

■ *usrplt*

Receives the electromagnetic force as an argument. This subroutine is called on completion of each calculation step, and can therefore output file in the required file format.

■ *usrsc*

The electric properties of superconductors can be arbitrarily defined. To make this subroutine available, select the Superconductor (user component) in the material electric properties.

■ *usrstr*

Define the stress dependent magnetizing property.

3-56-2. Environment Required for the User Subroutine

- C and Fortran are supported as the program language. Development environments where user subroutines can be built are the following.
  - C Language
    - Microsoft Visual Studio 2010
  - Fortran
    - Microsoft Visual Studio 2010 and Intel Fortran Composer XE 2011 (Intel Visual Fortran Compiler XE 12.1)

3-56-3. Building User Subroutines Using C Language

Before using the user subroutine, the subroutine in the user subroutine option must be modified (programmed) and built.

When the subroutine is used in Windows, the DLL (Dynamic Link Library) file is created as a result of the build. Unlike the static library, the DLL files can be located separately from the JMAG solver.

The DLL file can be used for any updated version of JMAG without any rebuild, so long as the interface of the user subroutine is the same.

3-56-3-1. Important Notes Regarding the Use of This Function

- Do not change the name of the file provided as the user subroutine, as well as the structure of the file that is extracted from usrsub_magnetic.zip.
- Do not edit the following files.
  - usrsub.cpp
  - usrsub.h

3-56-3-2. Microsoft Visual Studio 2010

1. Copy usrsub_magnetic.zip to any directory.

- Magnetic field analysis

  `(JMAG installation directory)\Solver\archive\usrsub_magnetic.zip`

- Thermal analysis

  `(JMAG installation directory)\Solver\archive\usrsub_heat.zip`

- Iron loss analysis

  `(JMAG installation directory)\Solver\archive\usrsub_ironloss.zip`

2. Extract the zip file.

3. Start Microsoft Visual Studio 2010.

4. Select [File] > [Open] > [Project] from the menu bar and open usrsub_cpp.sln from the following directory.

- Magnetic field analysis

  `(File-extracted-directory)\usrsub_magnetic\projects\vs2010\usrsub_cpp.sln`

- Thermal analysis

  `(File-extracted-directory)\usrsub_heat\projects\vs2010\usrsub_cpp.sln`

- Iron loss analysis

  `(File-extracted-directory)\usrsub_ironloss\projects\vs2010\usrsub_cpp.sln`

5. Open the C++ program file (*.cpp) on which the user subroutine is written from the following directory. Modify the required subroutine and save (overwrite) it.

- Magnetic field analysis

  `(File-extracted-directory)\usrsub_magnetic\src_cpp`

- Thermal analysis

  `(File-extracted-directory)\usrsub_heat\src_cpp`

- Iron loss analysis

  `(File-extracted-directory)\usrsub_ironloss\src_cpp`

6. Build the user subroutine DLL.

Select [Build] > [Build usrsub_cpp] (or [Build] > [Rebuild usrsub_cpp]) from the menu bar.

By running this menu command, a new DLL (default.dll) is created in the following directory.

- Magnetic field analysis

  `(File-extracted-directory)\usrsub_magnetic\build\vs2010\usrsub_cpp\Release`

- Thermal analysis

  `(File-extracted-directory)\usrsub_heat\build\vs2010\usrsub_cpp\Release`

- Iron loss analysis

  `(File-extracted-directory)\usrsub_ironloss\build\vs2010\usrsub_cpp\Release`

7. Place this DLL (default.dll) to the following directory.

- Windows
  - Magnetic field analysis

    `(JMAG installation directory)\usrsub\magnetic`

  - Thermal analysis

    `(JMAG installation directory)\usrsub\heat`

  - Iron loss analysis

    `(JMAG installation directory)\usrsub\ironloss`

- Linux
  - Magnetic field analysis

    `/works/solver/mod/magnetic/mod/usrsub/magnetic/`

  - Thermal analysis

    `/works/solver/mod/heat/mod/usrsub/heat/`

  - Iron loss analysis

    `/works/solver/mod/usrsub/ironloss/`

8. Specify a DLL file in a study property for JMAG-Designer, and in the analysis control condition for JMAG-Studio.

▶▶ Reference

- For more information about specifying the DLL file for magnetic field analysis, see P.323 "3-53. Controlling User Subroutine".
- For more information about specifying the DLL file for thermal analysis, see P.468 "4-27. Controlling User Subroutine".
- For more information about specifying the DLL file for iron loss calculation, see P.902 "12-3-9. Settings in the [User Subroutine] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)".

9. Run the analysis.

Important **When creating a user subroutine for 64 bit using VS2010, perform the following steps to set up the development environment.**

1. Open the project with VS2010.

2. Click the Solution Platform list box (labeled as Win32), and if available, select "x64" to enable the build configuration.

If "64 bit" does not appear in the list box, add it to the list box using the following steps.

3. Select the Configuration Manager from the Solution Platform list box.

4. Select New from the Active solution platform list.

5. Select x64 from the Enter or Select New Platform list box.

6. Select x64 from the Solution Platform list box.

Important **The build environment for Intel 64 needs to be added by selecting Custom Install when installing Visual Studio.**

Important **When installing Visual Studio and Intel Fortran on the same machine, make sure that they are installed in the correct order (first Visual Studio, and then Intel Fortran), and they are set up for building the 64bit module. For more information, see the manual of each toolkit.**

3-56-4. Building User Subroutines Using Fortran

Before using the user subroutine, the subroutine in the user subroutine option must be modified (programmed) and built.

When the subroutine is used in Windows, the DLL (Dynamic Link Library) file is created as a result of the build. Unlike the static library, the DLL files can be located separately from the JMAG solver.

The DLL file can be used for any updated version of JMAG without any rebuild, so long as the interface of the user subroutine is the same.

3-56-4-1. Important Notes Regarding the Use of This Function

- Do not change the name of the file provided at the user subroutine, and the structure of the file that is extracted from usrsub_magnetic.zip.
- The following unit numbers are already in use. Use the available unit numbers when the user subroutine is written in Fortran.

  Unit numbers in use:
  1, 2, 5, 6, 8, 10, 11, 12, 20, 21, 22, 23, 24, 25, 26, 27, 30, 40, 55, 56, 80, 82

3-56-4-2. Microsoft Visual Studio 2010+ Intel Visual Fortran

1. Copy usrsub_magnetic.zip to any directory.

- Magnetic field analysis
  `(JMAG installation directory)\Solver\archive\usrsub_magnetic.zip`

- Thermal analysis
  `(JMAG installation directory)\Solver\archive\usrsub_heat.zip`

- Iron loss analysis
  `(JMAG installation directory)\Solver\archive\usrsub_ironloss.zip`

2. Extract the zip file.

3. Start Microsoft Visual Studio 2010.

4. Select [File] > [Open] > [Project] from the menu bar to open usrsub_fortran.sln from the following directory.

- Magnetic field analysis
  `(File-extracted-directory)\usrsub_magnetic\projects\vs2010\usrsub_fortran.sln`
- Thermal analysis
  `(File-extracted-directory)\usrsub_heatc\projects\vs2010\usrsub_fortran.sln`
- Iron loss analysis
  `(File-extracted-directory)\usrsub_ironloss\projects\vs2010\usrsub_fortran.sln`

5. Open the Fortran program file (*.f) on which the user subroutine is written from the following directory. Modify the required subroutine and save (overwrite) it.

- Magnetic field analysis
  `(File-extracted-directory)\usrsub_magnetic\src_fortran`
- Thermal analysis
  `(File-extracted-directory)\usrsub_heat\src_fortran`
- Iron loss analysis
  `(File-extracted-directory)\usrsub_ironloss\src_fortran`

6. Build the user subroutine DLL.
Select [Build] > [Build usrsub_fortran] (or [Build] > [Rebuild usrsub_fortran]) from the menu bar.

By running this command, a new DLL (default.dll) is created in the following directory.

- Magnetic field analysis
  `(File-extracted-directory)\usrsub_magnetic\build\vs2010\usrsub_fortran\Release`
- Thermal analysis
  `(File-extracted-directory)\usrsub_heat\build\vs2010\usrsub_fortran\Release`
- Iron loss analysis
  `(File-extracted-directory)\usrsub_ironloss\build\vs2010\usrsub_fortran\Release`

7. Place this DLL (default.dll) to the following directory.

- Windows
  - Magnetic field analysis
    `(JMAG installation directory)\usrsub\magnetic`
  - Thermal analysis
    `(JMAG installation directory)\usrsub\heat`
  - Iron loss analysis
    `(JMAG installation directory)\usrsub\ironloss`
- Linux
  - Magnetic field analysis
    `/works/solver/mod/magnetic/mod/usrsub/magnetic/`
  - Thermal analysis
    `/works/solver/mod/heat/mod/usrsub/heat/`
  - Iron loss analysis
    `/works/solver/mod/usrsub/ironloss/`

8. Specify a DLL file in a study property for JMAG-Designer, and in the analysis control condition for JMAG-Studio.

▶▶ Reference

- For more information about specifying the DLL file for magnetic field analysis, see P.323 "3-53. Controlling User Subroutine".
- For more information about specifying the DLL file for thermal analysis, see P.468 "4-27. Controlling User Subroutine".
- For more information about specifying the DLL file for iron loss analysis, see P.902 "12-3-9. Settings in the [User Subroutine] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)".

9. Run the analysis.

Important **When creating a user subroutine for 64 bit using VS2010, perform the following steps to set up the development environment.**

1. Open the project with VS2010.

2. Click the Solution Platform list box (labeled as Win32), and if available, select "x64" to enable the build configuration.

If "64 bit" does not appear in the list box, add it to the list box using the following steps.

3. Select the Configuration Manager from the Solution Platform list box.

4. Select New from the Active solution platform list.

5. Select x64 from the Enter or Select New Platform list box.

6. Select x64 from the Solution Platform list box.

Important **The build environment for Intel 64 needs to be added by selecting Custom Install when installing Visual Studio.**

Important **When installing Visual Studio and Intel Fortran on the same machine, make sure that they are installed in the correct order (first Visual Studio, and then Intel Fortran), and they are set up for building the 64bit module. For more information, see the manual of each toolkit.**

3-56-5. User Subroutine Data in the Report File

When the user subroutine is used, the information on the user subroutine or the error message is included in the report file.

3-56-5-1. When the User Subroutine Runs Successfully

The following information is included in the report file.

- The path to the user subroutine (DLL file) in use
- The title of the user subroutine
- The version of the user subroutine
- Description
- Build date

The title, version, description and the build date are the ones written on src_cpp\info.cpp (C) or src_fortran\info.f (Fortran) available in the user subroutine file usrsub_magnetic.zip. By modifying these files while creating the user subroutine, the information above can be included in the report file.

3-56-5-2. When the User Subroutine Fail to Run

The following error / warning code may appear.

Error (Warning) ID	Message	Description
29052	Failed to load the user subroutine DLL	DLL is not found on the specified location, or the module not appropriate for DLL is found.
29053	Unable to find default.dll default.	dll is not found on the specified location.
20954	Unable to find the entry point	The user subroutine (function) may have been renamed while editing the C program file.

3-56-5-3. Generating Error Messages

Using the following functions, error/warning messages can be generated by a user subroutine.

■ C/C++

The argument is expressed in the same form as the print function.

- error: const* title, const char* format, etc.
- warning: const* title, const char* format, etc.
- message: const* title, const char* format, etc.

■ Fortran

- error: f_error(title,s)
- warning: f_warning(title,s)
- message: f_message(title,s)

Running the function of an error terminates the analysis. The warning message does not contain the information that the analysis is terminated. The title and contents of a message specified by these functions are included in the report file.

MEMO A frequent use of the functions above may slow the analysis due to the process of generating a message file.

3-56-6. Building a User Subroutine (Linux)

1. Copy the following files to a folder and then decompress them.

- Magnetic field analysis
  `/works/solver/mod/magnetic/usrsub/archive/usrsub_magnetic.tar.gz`

- Thermal analysis
  `/works/solver/mod/heat/usrsub/archive/usrsub_heat.tar.gz`

- Iron loss analysis
  `/works/solver/mod/tools/Loss/usrsub_ironloss.tar.gz`

2. Use the following command to decompress the files. (Shown below is for magnetic field analysis.)

```
gzip -d usrsub_magnetic.tar.gz
tar xvf usrsubdll_magnetic.tar
```

3. Create the code for the user subroutine using the source code saved in the following directories.

- C language
  `(File-extracted-directory)/usrsub_magnetic/src_cpp`

- Fortran
  `(File-extracted-directory)/usrsub_magnetic/src_frotran`

For thermal analysis, replace “gusrsub_magnetic” with “usrsub_heat”, and for iron loss analysis, replace it with “usrsub_ironloss”.

4. Move the current directories to the following directories.

- Magnetic field analysis
  `(File-extracted-directory)/usrsub_magnetic/projects/linux`

- Thermal analysis
  `(File-extracted-directory)/usrsub_heat/projects/linux`

- Iron loss analysis
  `(File-extracted-directory)/usrsub_ironloss/projects/linux`

5. Run the following commands.

- C language
  `make`

- Fortran
  `make fortran`

When the above commands are run, default.so is created in the following directories.

- C language
  `(File-extracted-directory)/usrsub_magnetic/build/linux/usrsub_cpp`

- Fortran
  `(File-extracted-directory)/usrsub_magnetic/build/linux/usrsub_fortran`

For thermal analysis, replace "usrsub_magnetic" with "usrsub_heat", and for iron loss analysis, replace it with "usrsub_ironloss".

6. Copy default.so to the following directories.

- Magnetic field analysis
  `/works/solver/mod/magnetic/mod/usrsub/magnetic`

- Thermal analysis
  `/works/solver/mod/heat/mod/usrsub/heat`

- Iron loss analysis
  `/works/solver/mod/usrsub/ironloss/`

MEMO To delete a created object file or module, use the following steps. 1. Move the current directory to the following directory.

1. Move the current directory to the following directory.
   Magnetic filed analysis
   (File-extracted-directory)/usrsub_magnetic/projects/linux
   Thermal analysis
   (File-extracted-directory)/usrsub_heat/projects/linux
   Iron loss analysis
   (File-extracted-directory)/usrsub_ironloss/projects/linux
2. Run the following command.
   make clean

Note that the module that is copied using step 6 P.343 "3-56-6. Building a User Subroutine (Linux)" above will not be deleted.

3-56-7. magusr

3-56-7-1. Settings

- Defines the magnetizing properties of magnets. This is used to describe anisotropy and nonlinearity of magnetizing properties.
  - For JMAG-Studio, to use this subroutine, select [User Subroutine] in the [Type] list in the [Magnetizing Properties] dialog box.
  - For JMAG-Designer, to use this subroutine, select [User Subroutine magusr, usrmtx] in the [Magnetic Properties] tab in the [Material Editor] dialog box.
- magusr is available in all magnetic field analysis modules.
- In *magusr*, return magnetization vector "*amu*" (A/m) and the rate of change in the magnetic flux density of magnetization vector "*dmdbu*" to JMAG.
- In particular, when magnetizing properties have nonlinearity, nonlinear iterations may not converge for wrong "*dmdbu*".
- "*amu*" and *"dmdbu*" correspond to M(B) and $\frac{\partial M}{\partial B}$ shown in the following equations, respectively.

$$H(B) = \frac{B}{\mu_0} - M(B)$$

$$\frac{\partial M}{\partial B} = \frac{1}{\mu_0} - \frac{\partial H}{\partial B}$$

- *magusr* is called by JMAG at the time when the following events occur.

  i. To define magnetizing properties for each element, the file *magusr*.f is called by the magnetic field analysis solver at each step of an iteration.

  "*buttond*", described later, is delivered with its value of 1. Return the magnetization vector "*amu*" and the differential term of magnetic flux density *"dmdbu*".

  ii. After the completion of nonlinear iterations, the file *magusr*.f is called by the magnetic field analysis solver in order to define magnetizing properties for each element for calculating magnetic field density and magnetic fields.

  "buttond" is delivered with its value of 2. Return the magnetization vector "*amu*".
- With this subroutine described, electromagnetic force and torque or accumulated energy cannot be calculated, as JMAG cannot define the energy. To determine the electromagnetic force, define the energy item in *usrene*.

3-56-7-2. Arguments

Variable	Details	I / O
time	Time (sec) or frequency (Hz)	I
istep	Step	I
buttond	Flag showing status of called subroutine: 1: In process of magnetic field analysis 2: In process of determining magnetic field or magnetization after magnetic flux analysis	I
nowelm	Element ID	I
cod (3)	Coordinates of element center (m)	I
tempr	Temperature (Celsius)	I
iflgm	ID flag of material property (negative integer)	I (<O)
bvec (3)	Magnetic flux at element center (T)	I
bveco (3)	Magnetic flux density at element center in the previous step (T)	I
amv (3)	Magnetization vector of magnet (A/m) (value set in magnet conditions)	I
bfac	Correction coefficient of flux density	I
numnp	Total number of nodes	I
numel	Total number of elements	I
maxnod	Maximum number of nodes within an element 2D analysis: 4 3D analysis: 8	I
ietyp (numel)	Element type	I
lm (maxnod, numel)	Node ID within an element	I
coord (3,numnp)	Node coordinates	I
iusprm	User-defined array size	I
usprm	User-defined array	I/O
amu (i)	Magnetization vector (A/m)	O
dmdbu (i, j)	Mj /Bi (symmetry matrix)	O
"I" is a variable inherited from JMAG and "O" is a value passed to JMAG.		

3-56-7-3. Example

This example represents magnetizing properties with anisotropies. B-H curves different by axis are applied, and the "*amu*" and "*dmdbu*" are determined.

■ Program (Fortran)

```
      subroutine magusr(buttond,istep,time,
     #                  nowelm,iflgm,cod,tempr,
     #                  bvec,bfac,bveco,amv,
     #                  numel,numnp,maxnod,ietyp,lm,coord,
     #                  iusprm,usprm,
c                       -- OUT
     #                  amu,dmdbu)
      implicit real*8 (a-h,o-z)
c     -- IN
      complex*16 bvec(3)
      dimension amv(3)
      dimension
cod(3),coord(3,numnp)ietyp(numel),lm(maxnod,numel)
      dimension  bveco(3),usprm(iusprm)
c     -- OUT
      complex*16 amu(3)
      dimension  dmdbu(3,3)
c     -- TMP
      dimension ntab(3)tabl(2,4,3)
      data ntab / 3,4,3 /
      data tabl /
     #  0.00000000000e+000, 0.00000000000e+000,
     #  7.95774715500e+003, 1.00000000000e+000,
     #  8.75352187000e+004, 2.00000000000e+000,
     #  0.00000000000e+000, 0.00000000000e+000,
     #  0.00000000000e+000, 0.00000000000e+000,
     #  3.97887357700e+003, 1.00000000000e+000,
     #  7.95774715500e+003, 1.50000000000e+000,
     #  4.77464829300e+004, 2.00000000000e+000,
     #  0.00000000000e+000, 0.00000000000e+000,
     #  7.95774715500e+003, 1.00000000000e+000,
     #  8.75352187000e+004, 2.00000000000e+000,
     #  0.00000000000e+000, 0.00000000000e+000 /
c
      pai = 3.14159265
      pmu0 = 4.0*pai*1.0d-7
c     -- initialization
      do 100 ii=1,3
        do 110 jj=1,3
          dmdbu(jj,ii) = 0.d0
 110    continue
        amu(ii) = cmplx(0.d0,0.d0)
 100  continue
c
      do 1000 ii=1,3
```

```
          absb = sqrt( bvec(ii)*dconjg(bvec(ii)) )
          do 1100 jj=2,ntab(ii)
            b1 = tabl(2,jj-1,ii)
            b2 = tabl(2,jj,ii)
            if(absb.ge.b1 .and. absb.le.b2)then
c             -- Find region
              h1 = tabl(1,jj-1,ii)
              h2 = tabl(1,jj,ii)
              vech = h1 + (h2-h1)*(absb-b1)/(b2-b1)
              vecm = absb/pmu0 - vech
              dmdb = 1/pmu0 - (h2-h1)/(b2-b1)
              goto 1110
            endif
 1100     continue
          b2 = tabl(2,ntab(ii),ii)
          h2 = tabl(1,ntab(ii),ii)
          vech = h2 + (absb - b2)/pmu0
          vecm = b2/pmu0 - h2
          dmdb = 0.d0
 1110     continue
          if(absb.gt.0.d0)then
            amu(ii) =  vecm*bvec(ii)/absb
          endif
          dmdbu(ii,ii) = dmdb
 1000   continue
c
        return
        end
```

3-56-8. conusr

3-56-8-1. Settings

- Defines the electric properties of conductors. This is used to describe aeolotropies of electrical conductivity.
- This subroutine is available in all magnetic field analysis modules except for static analysis.
- For normal electric conductors, *ctens* is passed from JMAG after it becomes zero.
- For superconductors, *ctens* is passed from JMAG after an initial electrical conductivity or an electrical conductivity in the previous nonlinear iteration which was determined from I-V or Bean characteristics is assigned in the diagonal member of TSIG, and 0 is assigned in the off-diagonal.
- Define *vsig* and *ctens*.
- For JMAG-Designer, to use this subroutine, select [User Subroutine conusr] in the [Electric Properties] tab in the [Material Editor] dialog box.
  For JMAG-Studio, to use this subroutine, select the [User Subroutine] radio button in the [Electric Properties] tab in the [Material] dialog box.

MEMO Calculating superconductors is only supported in JMAG-Studio.
For more information about superconductors, see "2-10. Superconductivity Properties (JMAG-Studio)" on page 64.

3-56-8-2. Arguments

Variable	Details	I / O
nowelm	Element ID	I
iflgc	Identification flag of material (negative integer)	I
matid	Material ID of an interesting element	I
cod (3)	Coordinate of element center (m)	I
ctens (3,3)	Electrical conductivity tensor (1/(ohm・m))	O
vsig	Electrical conductivity against current flow direction (1/(ohm・m))	O

- *vsig* is used only for loss calculation.

3-56-8-3. Example

This example represents electrical conductivity with anisotropies. The x and y components of electrical conductivity are set to 1000000.0 and the z component to 0.0.

■ **Program (Fortran)**

```
      subroutine conusr (nowelm,iflgc,matid,cod,
     #                           ctens,vsig)
      implicit Real*8 (a-h,o-x)
      complex*16 ctens(3,3)
      dimension cod(3)
c
      ctens(1,1) = cmplx(10000000.0,0.0)
      ctens(2,2) = cmplx(10000000.0,0.0)
      ctens(3,3) = cmplx(0.0,0.0)
c
      vsig = 10000000.0
      return
      end
```

3-56-9. usrfma

- Based on the electromotive force and torque calculated in an analysis, the distance and time interval can be defined for the next step.
- In the Motion condition, it can be used when the [User Subroutine] flag selected. In this case, input a negative integer into the text box.
- There are two ways to define a motional state.
  - Returning displacement to move to the specified time increment to a main routine Δt .
  - Returning the time increment necessary to displace the specified amount Δt to a main routine Δx .
- Specified displacement and rotation quantity, or time can be returned to JMAG.
- Modules in which this function can be used are transient response analysis (TR, DP).
- If you use *usrfma*, motion conditions and electromagnetic force and torque conditions must be set to the motional object.
- Motion conditions differ depending on whether *usrfma* are used. If you use *usrfma*, velocity of motion conditions must be zero.

usrfma.f	Motional state	Rotation center coordinates	Rotation center coordinates	Velocity
Use	Rotation motion	Mandatory	Mandatory	Set to 0 (round/sec)
Use	Translational motion	Unnecessary	Unnecessary	Set to 0 (m/sec)
Not use	Rotation motion	Mandatory	Mandatory	Mandatory
Not use	Translational motion	Unnecessary	Unnecessary	Mandatory

A motional state is defined by the following equation.

$$F - mg = ma + KV \qquad \text{(C-1)}$$

- *F*: force
- *m*: mass
- *a*: acceleration
- *K*: coefficient
- *g*: acceleration of gravity
- *V*: velocity

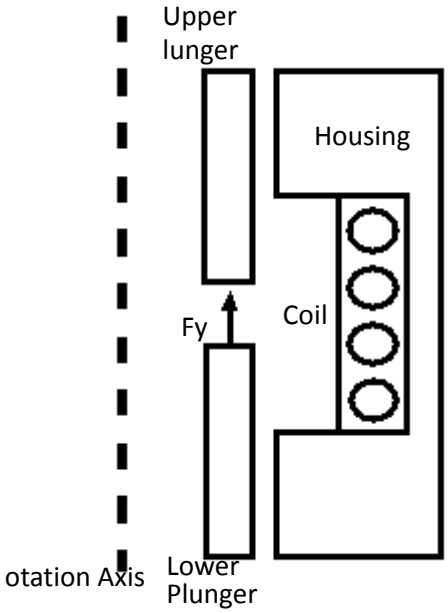
The velocity and acceleration are defined as:

$$V = \frac{\Delta x}{\Delta t} \qquad \text{(C-2)}$$

$$a = \frac{\Delta V}{\Delta t} \qquad \text{(C-3)}$$

- X: displacement

Where, x denotes the displacement. The following is a solenoid valve to be analyzed as an example. How the lower plunger is attracted by the upper plunger when applying a current to the coil is analyzed here.



Solenoid valve model

3-56-9-1. Specifying Displacement

Calculates the time necessary for an object to move the specified displacement and returns the time to the JMAG Solver.

- The analysis time is the calculated time regardless of the time input.
- In the Motion condition, it can be used when [User Subroutine] flag is selected. Specify 0 for the velocity or distance of the velocity point sequence of motion condition.
- With motion causing inconsistency in meshes, this can be set only in FEM+BEM analysis of DP (2D and axial symmetry magnetic field analysis (transient response)) module.

A sample program is shown below:

With k as the current step, equation (C-1) can be expressed as:

$$F^{k-1} - mg = m\frac{\Delta V^k}{\Delta t^k} + K\frac{\Delta x^k}{\Delta t^k} \quad \text{(C-a-1)}$$

Substituting $\Delta V^k = V^k - V^{k-1}$ and $V^k = \Delta x^k / \Delta t$ into the equation (C-a-1), we obtain

$$(F^{k-1} - mg)\Delta t = m\frac{\Delta x^k}{\Delta t} - mV^{k-1} + K\Delta x^k \quad \text{(C-a-2)}$$

Arrange the equation (C-a-2) for Δt

$$(F^{k-1} - mg)\Delta t^2 = (K\Delta x^k - mV^{k-1})\Delta t - m\Delta x^k = 0 \quad \text{(C-a-3)}$$

Hence we can solve the quadratic formula for Δt

$$\Delta t = \frac{K\Delta x^k - mV^{k-1} \pm \sqrt{(K\Delta x^k - mV^{k-1})^2 + 4(F^{k-1} - mg)m\Delta x^k}}{2(F^{k-1} - mg)} \quad \text{(C-a-4)}$$

Here, the time increment necessary to displace Δx^k is determined.

The program to represent the equation (C-a-4) is shown below:

■ Arguments

Variable	Details	I / O
instep	Current step	I
time1	Time of the previous step (sec)	I
time2	Time of current step (sec)	I
ngrpm	Summation of motion groups	I
nowgrp	Motion group ID to define	I
ngrpf	Summation of electromagnetic force group	I
oldF (4,ngrpf)	Electromagnetic force of the previous step	I
oldvl (4,ngrpm)	Velocity of the previous step	I
dispo (4,ngrpm)	Displacement until the previous step	I
oldac (4,ngrpm)	User-defined acceleration at the previous step Arbitrary setting is possible because it this value has not yet being determined in the JMAG.	I/O
dx	Displacement in X-axis (m/Step)	O
dy	Displacement in Y-axis (m/Step)	O
dz	Displacement in Z-axis (m/Step)	O
dr	Displacement in rotation axis (deg/Step)	O
vx	Velocity in X-axis (m/sec)	O
vy	Velocity in Y-axis (m/sec)	O
vz	Velocity in Z-axis (m/sec)	O
vr	Velocity in rotation axis (deg/sec)	O
iusprm	User-defined array size	I
usprm (iusprm)	User defined array	I/O

■ Example

It is assumed that an object moves toward the Y-axis direction. The mass of an object is assumed to be 1.015 kg, coefficient of friction 0.0, and acceleration of gravity 9.8 m/sec^2. Arguments are a little different with 2D analysis. For arguments, see example of (b) Specifying time.

■ Program (Fortran)

```
       subroutine usrfma(istep,time1,time2,ngrpm,nowgrp,
      #                      ktblm,ngrpf,oldF,oldvl,dispo,
      #                      oldac,
      #                      dx,dy,dz,dr,vx,vy,vz,vr,
      #                      usprm,iusprm)
       implicit real*8 (a-h,o-z)
       dimension dispo(4,ngrpm),oldvl(4,ngrpm), oldF(4, ngrpf)
       dimension usprm(iusprm) ,oldac(4,ngrpm)
       wght = 1.015
       frict = 0.0
       gravit = 9.8
       dtimex = 0.0d0
       eps000 = 1.0e-30
       eps00  = 1.0e-20
       do 100 i=1,4
c     In the case of movement toward Y-axis
          if(i.eq.2)then
c     Force is calculated with the following equation
            Force = oldF(i,nowgrp) - wght*gravit
c     Decision of divided by zero in equation (C-a-4)
            if(abs(Force).lt.eps000 .or. abs(dy).lt.eps00)then
            go to 100
            endif
c    Calculate the equation in the first parentheses in  in
equation  (C-a-4)
            rb = frict*dy - wght*oldvl(i,nowgrp)
c    Calculate the equation in   in equation (C-a-4)
            rc = rb * rb + 4.0d0 * Force * wght * rd
c    Calculate the time increment of current step
           dtimex = abs((rb + abs(sqrt(rc))) / (2.0d0 * Force)-mg)
            endif
            dtime = dtimex
      100  continue
c    Update the calculation time of current step
          time2 = time1 + dtime
c*******************************************************
       return
       end
```

3-56-9-2. Specifying Time

Calculates the displacement that an object moves during the specified time and returns the displacement to the JMAG Solver.

- Displacement during the specified time is calculated.
- In the Motion condition, it can be used when the [User Subroutine] flag is selected. Specify 0 for the velocity or distance of the velocity point sequence of motion conditions.
- In the TR (3D transient response magnetic field analysis) module, be sure not to cause inconsistency in meshes after movement
  A sample program is shown below:
  With k as the current step, equation (C-1) can be expressed as:

$$F^{k-1} - mg = m\frac{\Delta V^k}{\Delta t} + K\frac{\Delta x^k}{\Delta t} \quad \text{(C-b-1)}$$

Substituting $\Delta V^k = V^k - V^{k-1}$ and $V^k = \Delta x^k / \Delta t$ into the equation (C-b-1), we obtain

$$(F^{k-1} - mg)\Delta t = m\frac{\Delta x^k}{\Delta t} - mV^{k-1} + K\Delta x^k \quad \text{(C-b-2)}$$

Transforming the equation (C-b-2) for Δx^k, we obtain

$$\Delta x^k = \frac{\Delta t}{m + K\Delta t^k}\{(F^{k-1} - mg)\Delta t + mV^{k-1}\} \quad \text{(C-b-3)}$$

and, the displacement Δx^k is determined.
The program to represent the equation (C-b-3) is shown below:

■ Arguments

Variable	Details	I / O
instep	Current step	I
time1	Time of the previous step (sec)	I
time2	Time of current step (sec)	O
ngrpm	Summation of motion groups	I
nowgrp	Motion group ID to define	I
ngrpf	Summation of electromagnetic force group	I
oldF (4, ngrpf)	Electromagnetic force of the previous step	I
oldvl (4,ngrpm)	Velocity of the previous step	I
dispo (4,ngrpm)	Displacement until the previous step	I
oldac (4,ngrpm)	User-defined acceleration at the previous step This value has not yet being determined in the JMAG body.	I/O
dx	Displacement in X-axis (m/Step)	O
dy	Displacement in Y-axis (m/Step)	O
dz	Displacement in Z-axis (m/Step)	O
dr	Displacement in rotation axis (deg/Step)	O
vx	Velocity in X-axis (m/sec)	O
vy	Velocity in Y-axis (m/sec)	O
vz	Velocity in Z-axis (m/sec)	O
vr	Velocity in rotation axis (deg/sec)	O
iusprm	User-defined array size	I
usprm (iusprm)	User defined array	I/O

■ Example

It is assumed that an object moves toward the Y-axis direction. The mass of an object is assumed to be 1.015 kg, coefficient of friction 0.0, and acceleration of gravity 9.8 m/sec^2.

■ Program (Fortran)

```
      subroutine usrfma(istep,time1,time2,ngrpm,nowgrp,
      #                  ktblm,ngrpf,oldF,oldvl,dispo,
      #                  oldac,
      #                  dx,dy,dz,dr,vx,vy,vz,vr,
      #                  usprm,iusprm)
       implicit real*8 (a-h,o-z)
       dimension dispo(4,ngrpm),oldvl(4,ngrpm),oldF(4,ngrpf)
       dimension usprm(iusprm),oldac(4,ngrpm)
       wght = 1.015
       frict = 0.0
       gravit = 9.8
       eps00= 1.0e-20
c  Calculate t
       dtime = time2 - time1
c  Calculate the coefficient member on the right side of
equation (C-b-3)
       r = dtime/(wght+dtime*frict)
c  Calculate the expression in parentheses on the right side of
equation C-b-3)
       Force = oldF(2,nowgrp) - wght*gravit
c   Determine displacement dy at the current step in equation
(C-b-3)
       if(Force.gt.eps00)then
       dy = (oldvl(2,nowgrp)*wght+Force*dtime)*r
       else
       dy = 0.0d0
       endif
       dx = 0.0d0
       dz = 0.0d0
       dr = 0.0d0
       return
       end
```

3-56-10. usrfm2

3-56-10-1. Settings

- Allows for changing mass, attenuation constant and spring constant if parameter entries are used for equations of motion in the motion conditions.
- This subroutine is called for each motion condition at the start of each step.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer.

3-56-10-2. Arguments

Variable	Details	I / O
igrp	Motion condition ID	I
istep	Step number	I
time1	Time of the previous step (sec)	I
time2	Time of current step (sec)	I
ngrpm	Summation of motion conditions	I
dspin (4)	Initial position	I
dispo (4)	Accumulated displacement from the 1st step to the previous step	I
oldvl (4)	Velocity of the previous step	I
nfcgm	Referred electromagnetic force and torque condition ID	I
oldf (4)	Electromagnetic force (N) and torque (N·m) of the previous step	I
usprm (iusprm)	User defined array	I / O
iusprm	User-defined array size	I / O
amm	Mass (kg)	I / O
abb (4)	Attenuation constant (N/(m/sec))	I / O
akk (4)	Spring constant (N/m)	I / O
fc0in (4)	Constant load (N) (Nm)	I / O

3-56-10-3. Example 1

The spring constant varies with spring extension. The value of *dspin* indicates the initial compression or extension at the 1st step. The compression and extension of the spring at the current position can be represented by adding *dispo*, displacement from the initial position to this value.

Assume that *dspin* (2, *igrp*) = -1e-3, force in the positive direction of Y-axis is generated as the spring is compressed (extended) in the negative direction of the Y-axis by 1 mm at the initial position. Then if the displacement becomes *disp* (2, *igrp*) = 2e-3 at a certain step *istep*, force acts in the negative direction of the Y-axis as the spring is extended (compressed) in the positive direction of Y-axis by 1 mm. Since compression or extension depends on its interpretation, take care when creating a table as shown below.

In the following example, the spring is placed in the positive direction of the Y-axis of a movable object, if the value of *ydsp* is negative, the spring is supposed to be extended. In this example, extension or compression of spring is described in table (1, XX) and spring constant in table (2, XX). As the distance between points on the table is not interpolated, the spring constant is 1000 N/m at the 1e-11 mm position.

■ **Program (Fortran)**

```
      subroutine usrfm2(igrp,istep,time1,time2,ngrpm,
     #                  dspin,dispo,oldvl,
     #                  nfcgm,oldF,
     #                  iusprm,usprm, amm,abb,akk,fc0in)
      implicit real*8 (a-h,o-z)
      dimension dspin(4),dispo(4),oldvl(4)
      dimension oldF(4),fc0in(4),usprm(iusprm)
      dimension abb(4),akk(4)
      dimension table(2,4)
      data table(2,4)
     #             / -1.00e+10 , 1000 ,
     #                0.00      , 1000 ,
     #                1.00e-10 , 500  ,
     #                1.00e+10 , 500 /
      ydisp = coefm(2igrp)+ disp(2igrp)
       do 1000 ii=1,3
          if(ydisp.gt.table(1,ii) .and.
ydisp.le.table(2ii))then
             akk = table(2ii)
          endif
 1000 continue
      return
      end
```

3-56-10-4. Example 2

This shows the setting method used when the spring constant varies with the position of the rotor.

■ **Operation logic of usrfm2.f.:**

The use of equation of motion of motion conditions and selected the check box of [User Subroutine] in the [Analysis Control] dialog box (or property of study) always call *usrfm2*.

The flow is shown below.

1. Start analysis

2. Load analysis input data

3. Determine calculation time

4. Move rotor

i. Call *usrfm2*, and set weight, spring constant and attenuation constant. Values specified by analysis input data are used.

ii. Solve the equation of motion using the above variables

iii. Move the nodes of the rotor based on the displacement obtained

5. Solve the matrix of field

6. Export the results

7. If the calculation step is not the final step, return to 3.

In other words, although the spring constant can be changed with *usrfm2*, the value of electromagnetic force obtained by solving the equation of motion of 4-ii. will not feed back to 4-i. while calculating at the same time.

In the *usrfm2* shown in following example, when the rotor moves in the +Y direction, the spring constant can be changed according to its displacement. The referenced displacement is not the current time displacement obtained in 4-ii., but the previous time displacement determined in 4-ii.

About usrfm2.f.

The *akk* is a spring constant (N/m). The purpose of this subroutine is to determine this value. The table simulates the characteristics of a nonlinear spring. If the rotor does not move, *akk* is zero. When it moves in the + Y direction by 0.1 m, *akk* is 1000. When it moves in the +Y direction by 0.3m, akk is –1000 N/m. If the rotor is far from 0.4 m, *akk* is supposed to be zero.

Program (Fortran)

```
      subroutine usrfm2(igrp,istep,time1,time2,ngrpm,
     #                  dspin,dispo,oldvl,nfcgm,oldF,
     #                  iusprm,usprm,amm,abb,akk,fc0in)
      implicit real*8 (a-h,o-z)
      dimension dspin(4),dispo(4),oldvl(4)
      dimension oldF(4),fc0in(4),usprm(iusprm)
      dimension abb(4),akk(4)
      parameter (ntab = 6)
      dimension table(2,ntab)
      data table / 0.0000000E+00,     0.0,
     #             1.0000000E-01,       1000.0,
     #             2.0000000E-01,       0.0,
     #             3.0000000E-01,       -1000.0,
     #             4.0000000E-01,       0.0 ,
     #             1.0000000E+10,       0.0 /
C EXAMPLE Y displacement vs spring constant
C     table(1,mm) : displacement Y   (m)
C     table(2mm) : spromg constant  (kg/m)
C     -- Set table value(X)
      dispY = dspin(2)+dispo(2)
C     -- Set number of table column
      if(dispY.lt.table(1,1))then
        valk = table(2,1)
      elseif(dispY.gt.table(1,ntab))then
        valk = table(2,ntab)
      else
        do 1000 ip=2,ntab
          ip1 = ip - 1
          ip2 = ip
          if(dispY.ge.table(1,ip1) .and.
     #       dispY.le.table(1,ip2)       )then
            dy = table(1,ip2) - table(1,ip1)
            dk = table(2,ip2) - table(2,ip1)
            ddy = dispY - table(1,ip1)
            ddk = dk*ddy/dy
            valk = table(2,ip1) + ddk
            go to 10
          endif
 1000   continue
 10     continue
      endif
```

```
akk = valk
return
end
```

3-56-11. usrcir

3-56-11-1. Settings

- To make this subroutine available, use the user subroutine type of circuit component.
- This is available all magnetic field analysis modules except static analysis (ST).

3-56-11-2. Arguments

Variable	Details	I / O
igrp9	User variable	I
istep	Step number	I
time	Time (sec)	I
voltr	Voltage (real part) (V)	I
volti	Voltage (imaginary part) (V)	I
currr	Current (real part) (A)	I
curri	Current (imaginary part) (A)	I
igrl9	Component type	O (Mandatory, any of 4 types)
		=-1 Resistor
		=-2 Capacitor
		=-3 Coil
		=-4 Switch
zimp1	Property value of component	O (Mandatory except for switch)
iswch	Open/close state of component	O (Mandatory, any of 3 types)
		=-1 Component other than switch
		=0 Switch: Off
		=1 Switch: On

3-56-11-3. Example

Properties of circuit components are defined based on the user variable igrp9 that was assigned when setting circuit components types. In this example, four types of user defined circuit components are defined individually.

Four types of components are available; resistors, capacitors, coils, and switches. The component is a resistor with igrp9 value 1, capacitor with 2, coil with 3, and switch with 4.

The switch is closed until the time of 0.0002 sec passes, and then opens.

■ **Program (Fortran)**

```
       subroutine usrcir(igrp9,
C                                -- IN
     #                           istep,time,voltr,volti,
     #                           currr,curri,
C                                -- OUT
     #                           igrl9,zimp1,iswch)
      implicit real*8 (a-h,o-z)
      if(igrp9.eq.1)then
        igrl9 = -1:Resistor
        zimp1 = 5.d0:5ohm
        iswch = -1
      else if(igrp9.eq.2)then
        igrl9 = -2:Capacitor
        zimp1 = 1e-6:1e-6 (F)
        iswch = -1
      else if(igrp9.eq.3)then
        igrl9 = -3:Coil
        zimp1 = 0.02:0.02H
        iswch = -1
      else if(igrp9.eq.4)then
        igrl9 = -4:Switch
        zimp1 = 0.d0
        if(time.ge.0.0002)then
          iswch = 0:Switch is open
        else
          iswch = 1:Switch is closed
        endif
      endif
      return
      end
```

3-56-12. usrstp

3-56-12-1. Settings

- Allows for making forced changes of step and time. The use of this subroutine means that analysis is performed with analysis input data such as time being overridden.

  You should not use this in normal analysis.
- This is called only once at the start of each analysis step.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer.

3-56-12-2. Arguments

Variable	Details	I / O
istep	Step number	I
time	Time (sec)	I / O

3-56-12-3. Example

In this example, even if you increase the calculation steps, the time is always 0 sec in the analysis.

■ **Program (Fortran)**

```
subroutine usrstp(istep,time)
implicit real*8 (a-h,o-z)
time = 0.d0
return
end
```

3-56-13. usrfle

3-56-13-1. Settings

- Files to be used can be opened with this subroutine.
- This subroutine is called once each at the start and end of analysis.
- Since the flag *mstat* is 0 at the start and 1 at the end, it can be used to determine whether to open or close a file.

  You should not use the reserved device numbers as indicated in "Notes on the use of user subroutines" when opening a file.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer.

3-56-13-2. Arguments

Variable	Details	I / O
mastat	Flag to determine analysis state, start or end	I
		=0:Start of analysis
		=1:End of analysis

3-56-13-3. Example

In this example, the file "jlocal" is opened with the device number of 102 at the start of analysis. The file is closed at the end of analysis.

■ **Program (Fortran)**

```
      subroutine usrfle(mstat)
      implicit real*8 (a-h,o-z)
C EXAMPLE
     if(mstat.eq.0)then
       open
(102,file='jlocal',recl=36,form='unformatted',access='direct')
     elseif(mstat.eq.1)then
       close(102)
     endif
      return
      end
```

3-56-14. usrxv

3-56-14-1. Settings

- It is available only in the JMAG Linux module.
- Allows for creating X-axis values that are needed for determining Y-axis values in a point sequence.
- At the time of reading the Y-axis value from the point sequence, if the user subroutine is specified as X-axis type, it will be called out.

3-56-14-2. Arguments

Variable	Details	I / O
istep	Step number	I
time	Time (sec)	I
iflag	Flag for X-axis type of point sequence	I
xpara	X-axis value	O

3-56-14-3. Example

In this example, the same value as the step number is returned to JMAG as X-axis value.

■ Program (Fortran)

```
subroutineusrxv(istep,time,iflag,xpara)
       implicit real*8 (a-h,o-z)
     if(istep.eq.1)then
       xpara = 1.d0
     elseif(istep.eq.2)then
       xpara = 2.d0
     elseif(istep.eq.3)then
       xpara = 3.d0
     elseif(istep.eq.4)then
       xpara = 4.d0
     else
       xpara = 5.d0
     endif
     return
     end
```

3-56-15. usrlos

3-56-15-1. Settings

- Use when the calculated Joule loss is to be modified.
- The Joule loss density calculated for each element is delivered as "closs". Add a correction value to the Joule loss and return it.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer.
- When calculation of each step is finished, and immediately after Joule loss has been calculated, they will be called out for each element.

3-56-15-2. Arguments

Variable	Details	I / O
istep	Step number	I
time	Time (sec)	I
ne	The element ID of the element focused on at present.	I
bvec (3)	Magnetic flux density (T)	I
bveco (3)	Magnetic flux density of the previous step (T)	I
eddy (3)	Current density (T)	I
numnp	Total number of nodes	I
numel	Total number of elements	I
maxnod	Maximum number of nodes in element	I
numat	Total number of physical properties	I
iel	Total number of physical properties	I
lm (i, nelm)	Node ID of 1st node of element ID NELM	I
coord (3, nnod)	Coordinate of node ID NNOD (m)	I
matid	Material ID of an interesting element	I
iflgm	Material property type flag of an interesting element	I
ntabm	Point number of material property point sequence of interesting elements	I
tablm (2, ntabm)	B (T) -M (A/m) curve if an interesting element is linear material	I
cmu	Permeability if an interesting element is linear material	I
tempr	Temperature	I
iusprm	User-defined array size	I

Variable	Details	I / O
usprm	User-defined array	I
bfac	Correction coefficient of magnetic flux density	I
closs	Joule loss density (W/m^3)	I / O

- *bvec*: Magnetic flux density (T). If the local coordinate system is used, the vector is given in the local coordinate system (U, V, W).
- *bveco*: Magnetic flux density (T) at the previous step. If the local coordinate system is used, the vector is given in the local coordinate system (U, V, W).
- *eddy*: Current density (A/m^2). If the local coordinate system is used, the vector is given in the local coordinate system (U, V, W).
- *closs*: Set the loss density (W/m^3). The Joule loss density determined by JMAG is already assigned this value, and it should therefore be noted down when overwriting.

3-56-15-3. Example

In this example, the iron loss, Ploss, is supposed to be zero.

■ Program (Fortran)

```
      subroutine usrlos(istep,time,ne,
     #                  bvec,bveco,eddy,
     #                  numnp,numel,maxnod,numat,
     #                  iel,lm,coord,
     #                  matid,iflgm,ntabm,tablm,cmu,
     #                  tempr,
     #                  iusprm,usprm,bfac,
c                       -- OUT
     #                  closs)
      implicit real*8 (a-h,o-z)
            complex*16 bvec(3),eddy(3)
        dimension  bveco(3)
           dimension lm(maxnod,numel),coord(3,numnp)
           dimension tablm(2*ntabm)
           dimension usprm(iusprm)
           complex*16 cmu
      Ploss = 0.d0
      closs = closs + Ploss
      return
      end
```

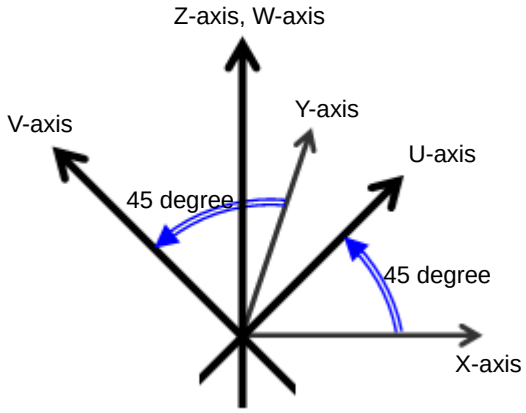
3-56-16. usrmtx

3-56-16-1. Settings

- This is used when the definition of the local coordinate system referenced in the steel sheets varies with the location of each element.
- The definition of the local coordinate system can be set for each element.
- This is called before user subroutines such as *magusr* if the number of the local coordinate system for the steel sheets is negative.
  The physical quantities (such as magnetic flux density) set in *magusr* are passed on.
- For JMAG-Studio, to use this subroutine, select [User Subroutine] in the [Anisotropy/Lamination Direction] dialog box.
- For JMAG-Designer, to use this subroutine, select [User Subroutine magusr, usrmtx] in the [Magnetic Properties] tab in the [Material Editor] dialog box.

3-56-16-2. Arguments

Variable	Details
ne	Element ID of element of interest
mtxtp	Local coordinate system number set for above element. (negative number)
iel	Number of nodes in element
xyz	Array of node coordinates in element. Units are (m).
csmt0	Local coordinate system. Define array
csmt0 (1,1)	X coordinate of origin O in local coordinate system
csmt0 (2,1)	Y coordinate of origin O in local coordinate system
csmt0 (3,1)	Z coordinate of origin O in local coordinate system
csmt0 (1,2)	X coordinate of a point A on axis U in local coordinate system
csmt0 (2,2)	Y coordinate of a point A on axis U in local coordinate system
csmt0 (3,2)	Z coordinate of a point A on axis U in local coordinate system
csmt0 (1,3)	X coordinate of a point B on axis V in local coordinate system
csmt0 (2,3)	Y coordinate of a point B on axis V in local coordinate system
csmt0 (3,3)	Z coordinate of a point B on axis V in local coordinate system



3-56-16-3. Example

■ Program (Fortran)

```
      subroutine usrmtx(ne,mtxtp,iel,xyz,
C                               -- OUT
     #                          csmt0)
      implicit real*8 (a-h,o-z)
      dimension xyz(3,8)
      dimension csmt0(3,3)
      csmt0(1,1) =  0.d0
      csmt0(2,1) =  0.d0
      csmt0(3,1) =  0.d0
      csmt0(1,2) =  1.d0
      csmt0(2,2) =  1.d0
      csmt0(3,2) =  0.d0
      csmt0(1,3) = -1.d0
      csmt0(2,3) =  1.d0
      csmt0(3,3) =  0.d0
      return
      end
```

3-56-17. hsusr1

3-56-17-1. Settings

- This subroutine generates a file that contains the information of each element. The file can be referred by the other subroutine for magnetizing property and electrical property.
- This subroutine is executed only once at the start of analysis.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer.

3-56-17-2. Arguments

Variable	Details	I/O
numel	Total number of elements	I
numat	Material ID for each element	I
matid (numat)	Material ID	I
mtyp (numel)	Material order ID for each element	I
iflgm (3,numat)	Material type for each element	I
iusprm	User-defined array size	I
usprm	User-defined array	I / O

3-56-17-3. Example

In this example, a temporary file jhist is created. To acquire the Material ID of each material, write "*matid*" (*mtyp*: Element ID).

■ **Program (Fortran)**

```
      subroutine hsusr1(numel, numat, matid, mtyp,iflgm,
     #                      iusprm,usprm)
      implicit real*8 (a-h,o-z)
      dimension matid(numat), mtyp(numel),iflgm(3,numat)
      dimension usprm(iusprm)

open(101,file='jhist',recl=36,form='unformatted',access='direc
t')
       zero=0.0
       izero=0
       ione=1
       do 1000 ne=1,numel
         write(101,rec=ne) zero,zero,zero,ione,ione,ione
 1000 continue
       return
       end
```

3-56-18. hsusr3

3-56-18-1. Settings

- The values of user-defined arrays can be changed on completion of calculations at each step or when the plot file is exported.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer. This is called when calculations and the plot file output are complete. It can be used only once for each step.

3-56-18-2. Arguments

Variable	Details	I/O
ne	Element ID of element of interest	I
matid	Material ID	I
iflgm (3)	Type of magnetizing properties	I
bvec	Current flux density (T)	I
bveco	Flux density at the previous step (T)	I
bvtr	Current flux density (T). Vector value represented in local coordinate system	I
bvotr	Flux density at the previous step (T). Vector value represented in local coordinate system	I
bfac	Variable used to multiply the flux density obtained by analysis in the post process. bfac should be set by subroutine *hsusr4*.f.	I
numel	Total number of elements	I
iusprm	User-defined array size (This should be defined in the analysis control.)	I
usprm	User-defined array. (This should be redefined.)	I / O

3-56-18-3. Example

■ **Program (Fortran)**

```
      subroutine hsusr3(ne,matid,numat,iflgm,
     #                  bvec,bveco,bvtr,bvotr,bfac,numel
c                       -- IN & OUT
     #                  iusprm,usprm)
      implicit real*8 (a-h,o-z)
c     -- IN
      dimension  iflgm(3)
      complex*16 bvec(3) , bvtr(3)
      dimension  bveco(3), bvotr(3)
c     -- IN&OUT
      dimension  usprm(iusprm)
C=========--------=========---------=========----------
      return
      end
```

3-56-19. hsusr4

3-56-19-1. Settings

- The correction coefficient of flux density, bfac, can be defined.
- This correction coefficient is applied only to the flux density distribution calculated at each step when it is exported. Therefore, if external circuits are used, the electric current flowing through or potential across each circuit component cannot be affected, and even if the output value of the flux density at a certain point is reduced by the correction coefficient, the reduced flux cannot flow into other regions.

  The following shows how the correction coefficient works on output items.
  - Output items with parameters that are proportional to the correction coefficient

    Joule loss density, magnetic flux, magnetization, magnetic field, and analysis error.
  - Output items with parameters that are exponentially proportional to the correction coefficient

    Joule loss, accumulated energy, electromagnetic force and torque, nodal force, surface force, lorentz force, and hysteresis loss (frequency response analysis only).
  - Output items independent from the correction coefficient

    Component electric potential of external circuits, electric current values of external circuits, electric current density set by the current density condition, electric current values set by the current condition, electric current density set by the FEM coil condition, flux linkage of FEM coil, and flux linkage in the current condition region
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer.
- This subroutine will be called once for each element immediately after magnetic flux density distribution is calculated and the calculation of each step converges. In other words, at the time calculation converges for Step 1, the same number of subroutines as the number of elements will be called from JMAG, and at the time the calculation converges for Step 2, the same number as the number of elements will be called again.

3-56-19-2. Arguments

Variable	Details	I/O
ne	Element ID of element of interest	I
matid	Material ID of element of interest	I
iflgm (3)	Type of magnetizing properties	I
bvec (3)	Flux density (T). Vector value on local coordinate system (U, V, W) if coordinate system is used.	I
bveco (3)	Array of flux density (T) of element at previous step. Vector value on local coordinate system (U, V, W) if coordinate system is used.	I
bfac	Variable array used to multiply the flux density obtained from analysis in the post process. Define the value pertaining to the element of interest. If a value other than 1 is set, the calculated value is corrected. When trying to correct the flux density to process magnetizing anisotropic, the correction coefficient should be defined using this subroutine. Various physical quantities obtained from calculation are multiplied by the correction coefficient.	O

3-56-19-3. Example

■ Program (Fortran)

```
      subroutine hsusr4(ne,
matid,iflgm,bvec,bveco,iusprm,usprm,
c                            -- OUT
     #                        bfac)
      implicit real*8 (a-h,o-z)
c           -- IN
      dimension  bveco(3),iflgm(3),usprm(iusprm)
      complex*16 bvec(3)
      bfac = 1.d0
C
      return
      end
```

3-56-20. usrhys

3-56-20-1. Settings

- Calculates the hysteresis loss generated for each element.
- It can be used when the check box of this subroutine is selected on the analysis control condition in JMAG-Studio, or on the property of study if using JMAG-Designer, and the [User Subroutine] flag is selected of the material setting.
- Upon completion of calculation, immediately after the magnetic flux density is calculated, a subroutine will be called for each element in each Step.

3-56-20-2. Arguments

Variable	Details	I/O
numel	Total number of elements	I
ne	Element ID of element of interest	I
bvec (3)	Current flux density vector (T). Vector value on local coordinate system (U, V, W) is passed on if coordinate system is used.	I
bveco (3)	Flux density vector (T) at previous step. Vector value on local coordinate system (U, V, W) is passed on if coordinate system is used.	I
iusprm	Size of user-defined array, *usprm*.	I
usprm	User-defined array.	I / O
bfac	Correction coefficient for flux. To be defined by hsusr4.f.	I
hloss	Hysteresis loss (W/m^3). Value should be defined.	O

3-56-20-3. Example

■ **Program (Fortran)**

```
      subroutine usrhys(numel,ne,bvec,bveco,
      #                  iusprm,usprm,bfac,
C                        -- OUT
      #                  hloss)
       implicit real*8 (a-h,o-z)
       complex*16 bvec(3)
       dimension  bveco(3)
       dimension  usprm(iusprm)
C
       hloss = 0.d0
C
       return
       end
```

3-56-21. usrene

3-56-21-1. Settings

- When determining each accumulated energy or electromagnetic force, energy values are required for each element. This routine enables determination of the energy and flux density, and the magnetic field.

  $$energy = \int HdB$$

- This subroutine is called by each element having the [User Subroutine] radio button depressed in the Material setting.
- This subroutine is called at the time when electromagnetic force and stored energy are obtained after completing calculation for each step.

3-56-21-2. Arguments

Variable	Details	I/O
numel	Number of all elements	I
ne	Element ID	I
matid	Material ID of element	I
iflgm(3)	Type of magnetizing properties	I
bvec(3)	Flux density (T) of element with element ID. Vector value on local coordinate system (U, V, W) is passed on if coordinate system is used.	I
bveco(3)	Flux density (T) of this element at previous step. Vector value on local coordinate system (U, V, W) is passed on if coordinate system is used.	I
amv(3)	Magnetization vector of this element (A/m). Vector value on local coordinate system (U, V, W) is passed on if coordinate system is used.	I
iusprm	Size of user-defined array (This is defined in the analysis control.)	I
usprm	User-defined array.	I / O
bfac	Flux density correction coefficient of this element. bfac should be defined in hsusr4.f. Do not set here.	I
energy	Energy value. Units are (J/m^3).	O
hh1, hh2, hh3	Assign magnetic field (X, Y, Z) and return value. The unit is (A/m).	O
bb1, bb2, bb3	Assign flux density (X, Y, Z) and return value. Units are (T).	O

MEMO The specifications of the variables of this subroutine (Iflgm) were changed as follows in the process of upgrading the versions of JMAG- Studio.
In the case of version 7X: Only one component is defined.
In the case of version 8X: Three components are defined.

3-56-21-3. Example: Linear Material with the Relative Permeability of 100 (TR)

■ Program (Fortran)

```
      subroutine usrene(numel,ne, matid,iflgm,bvec,bveco,amv,
     #                      iusprm,usprm,bfac,
C                          -- OUT
     #                      energy,hh1,hh2,hh3,bb1,bb2,bb3)
      implicit real*8 (a-h,o-z)
C
      complex*16 bvec(3)
      dimension  amv(3), bveco(3)
      dimension  usprm(iusprm)
      dimension  iflgm(3)
      cmu = 100.d0*4.0*3.1415926536*1e-7
      bb1 = bvec(1)
      bb2 = bvec(2)
      bb3 = bvec(3)
      hh1 = bvec(1)/cmu
      hh2 = bvec(2)/cmu
      hh3 = bvec(3)/cmu
      energy = 0.5.d0*(bb1*hh1+ bb2*hh2+ bb3*hh3)
      return
      end
```

3-56-22. usrplt

3-56-22-1. Settings

- Outputs electromagnetic force in desired format.
- This is called on completion of each calculation step.

3-56-22-2. Arguments

Variable	Details	I/O
nstep	Total calculation step	I
istep	Step	I
time	Current time (sec)	I
numnp	Total number of nodes	I
ngrpf	Electromagnetic force and torque conditions ID	I
ftotl	X-direction electromagnetic force (N)	I
torqe	Y-direction electromagnetic force (N)	I
nmoutf	The number of nodes of the elements that are the electromagnetic force and torque conditions are set	I
noutf	The nodal numbers of the nodes that electromagnetic force and torque conditions are set	I
frc_t	The nodal force (N) (The values of the nodes that do not have the electromagnetic force and torque conditions set, are also included.)	I

3-56-22-3. Example

■ **Program (Fortran)**

```
      subroutine usrplt(nstep,istep,time,
C                              -- Geometry
     #                         numnp,
C                              -- Force & Torque  (step)
     #                         ngrpf,ftotl,torqe,
C                              -- Force
     #                         nmoutf,noutf,frc_p)
      implicit real*8 (a-h,o-z)
      dimension noutf(nmoutf)
      complex*16 frc_p(3,numnp),ftotl(3,ngrpf),torque(ngrpf)
C
      return
      end
```

3-56-23. usrsc (JMAG-Studio)

3-56-23-1. Settings

- The current and electrical field properties of a superconductor can be set.
- This subroutine is called by each element having [Superconductor (User Component)] radio button depressed in the [Electric Properties] group box of the [Material] dialog box.
  Since the current density will be handed over from JMAG, define the electric field.

3-56-23-2. Arguments

Variable	Details	I/O
ne	Element ID of element of interest	I
center	Coordinates of element center (m)	I
vecj	Current density (A/m^2)	I
time	Time (sec)	I
temp	Temperature (Celsius)	I
vecE	Electric field (V/m)	O
vecoE	Electric field (V/m) at the previous step	I

MEMO The variable vecoE of this subroutine was added to version 8.1.1a or later of the magnetic field analysis solver (jam.exe).
The version of the magnetic field analysis solver can be verified by selecting [Help] > [About JMAG-Studio] from the menu bar, and clicking the [Details] button.

3-56-23-3. Example

■ Program (Fortran)

```
      subroutine usrsc(ne,center,vecJ,
     #                 time,temp, vecoE,
c                      -- OUT
     #                 vecE)
      implicit real*8 (a-h,o-z)
      dimension center(3),vecJ(3),vecE(3),vecoE(3)
      sigma = 1e7
      vecE(1) = vecJ(1)/sigma
      vecE(2) = vecJ(2)/sigma
      vecE(3) = vecJ(3)/sigma
      return
end
```

3-56-24. usrstr

Reference

For more information about the stress-dependent magnetizing properties, see P.29 "2-3-5. Specifying the Magnetic Properties of a Stress Dependency".

3-56-24-1. Settings

- Define the stress dependent magnetizing property.
- usrstr is called when:

1. **Each nonlinear iterative calculation is run to define the magnetizing properties for each element.**
   buttond (listed below) is set to “1”. Enter a magnetization vector, amu, and a derivative term of the flux density.
2. **The nonlinear iterative calculation has been completed, to define the magnetizing properties for each element required for calculating magnetic field.**
   buttond is set to “2”. Enter a magnetization vector, amu.

3-56-24-2. Arguments

Variable	Details	I/O
idusr	ID for a user subroutine	I
matid	Material ID of the stress-dependent magnetizing properties	I
buttond	Flag that indicates when a user subroutine is called (I) 1: Running magnetic field analysis 2: Processing the analysis results	I
istep	Step	I
time	Time (sec) or frequency (Hz)	I
bvec	Flux density at the center of element	I
pstrs	Principal stress (I), principal stress (II), principal stress (III) (Pa)	I
shstr	Shear stress, XY component, YZ component, ZX component (Pa)	I
strs3	Stress, X component, Y component, Z component (Pa)	I
numstr	Number of principal stress values included in hbs	I
maxbh	Maximum number of points in the B-H table	I
stbl	Principal stress (III) included in hbs (Pa)	I
sbh	Principal stress (III) included in hbs (Pa)	I
bhtbl	B-H table	I
iusprm	Size of user array	I
usprm	User array	I/O
amu	Magnetization vector (A/m)	O
dmdbu	∂ Mj / ∂ Bi (symmetric matrix)	O

3-56-24-3. Example

■ **Program (Fortran)**

```
c
      subroutine usrstr(idusr,buttond,istep,time,bvec,
     #                  pstrs,shstr,strs3,
     #                  numstr,maxbh,sttbl,sbh,bhtbl,
     #                  iusprm,usprm,
c                       -- OUT
     #                  amu,dmdbu)
      implicit real*8 (a-h,o-z)
c     -- IN
      dimension  pstrs(3),shstr(3),strs3(3)
      dimension
sttbl(numstr),sbh(numstr),bhtbl(2,maxbh,numstr)
      dimension  usprm(iusprm)
complex*16 bvec(3)
c     -- OUT
      complex*16 amu(3)
      dimension  dmdbu(3,3)
c*******************************************************
      pai  = 4.d0*atan(1.d0)
      pmu0 = 4.d0*pai*1.0d-7
c     -- initialization
      do 100 ii=1,3
        do 110 jj=1,3
          dmdbu(jj,ii) = 0.d0
 110    continue
        amu(ii) = dcmplx(0.d0,0.d0)
 100  continue
c
      absb = sqrt( bvec(1)*dconjg(bvec(1))
     #           + bvec(2)*dconjg(bvec(2))
     #           + bvec(3)*dconjg(bvec(3)) )
      do 1000 ii=2,numstr
        dstr = pstrs(3)-sttbl(ii)
        if(dstr.le.0.d0) then
          ip1=ii-1
          ip2=ii
          ps1=sttbl(ip1)
          ps2=sttbl(ip2)
          do 1100 jj=2,sbh(ip1)
            h1  = bhtbl(1,jj-1,ip1)
            h2  = bhtbl(1,jj,ip1)
            b11 = bhtbl(2,jj-1,ip1)
            b12 = bhtbl(2,jj,ip1)
            b21 = bhtbl(2,jj-1,ip2)
            b22 = bhtbl(2,jj,ip2)
            b1 = b21 + (b11-b21)*((pstrs(3)-ps2)/(ps1-ps2))
            b2 = b22 + (b12-b22)*((pstrs(3)-ps2)/(ps1-ps2))
            if(absb.ge.b1 .and. absb.le.b2)then
```

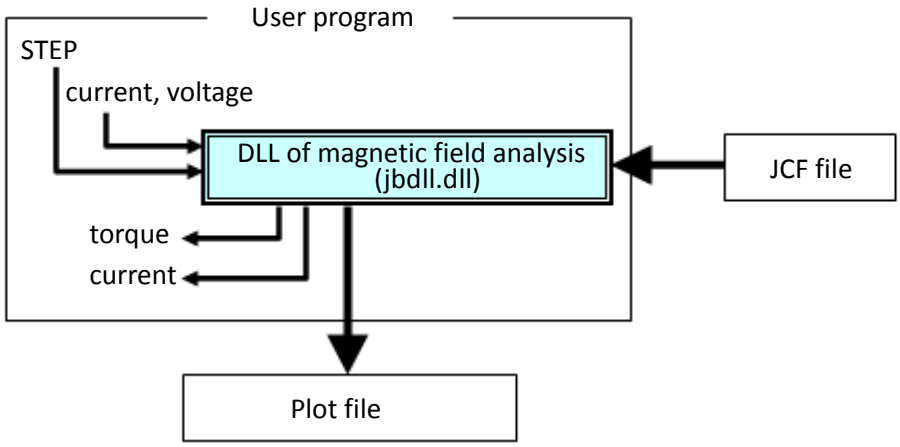
```
              vech = h1 + (h2-h1)*(absb-b1)/(b2-b1)
              vecm = absb/pmu0 - vech
              dmdb = 1.d0/pmu0 - (h2-h1)/(b2-b1)
              goto 1110
            endif
1100      continue
        endif
1000  continue
1110  if(absb.gt.0.d0)then
        amu(1) = vecm*bvec(1)/absb
        amu(2) = vecm*bvec(2)/absb
        amu(3) = vecm*bvec(3)/absb
      endif
      dmdbu(1,1) = dmdb
      dmdbu(2,2) = dmdb
      dmdbu(3,3) = dmdb

      return
      end
```

3-57. DLL of Magnetic Field Analysis (jbdll.dll)

3-57-1. Function

DLL enables magnetic field analysis to run from an external program.

Custom-made programs as well as commercially available software such as MATLAB/Simulink can be used as the external program. This can also be combined with personal programs.



JMAG module library, “jbdll.dll”, is a dynamic link library (DLL) designed to execute JMAG magnetic field analysis module from an external program created using C++.

The class, JMAG_JAM, is described in the DLL. This class has a function that executes a magnetic field analysis. Parameters such as calculation conditions can be changed because calculation information is retained in the memory during the execution of magnetic field analysis and the memory is freed at the end of execution.

The files required for compiling using this library are shown in table below. The files required for performing calculation are show in Table 2. The “jbdll.dll” provides classes. If C++ classes cannot be handled directly, for example, in compiling using C, you may have to create an intermediate program in C to handle “jbdll.dll” classes.

List of files required for compiling

File name	Description
jbdll.lib	Magnetic field analysis dynamic link library
jmag_jam.h	Header file for calculation control class, jmag_jam

List of files required during calculation

File name	Description
jinput	Magnetic field analysis input file (Solver file)
do_dpl.bat	Batch file

- When designating input data name for magnetic field analysis, use the function Set_SolverFileName discussed later on.
  When this function is not used, use the fixed name "jinput". After calculation, the file named "jplot" is output. If you modify the file name with the extension "plot", you will be able to view the results with JMAG-Studio. Most conditions that can be modified by "jbdll.dll" are those that can be modified only when Constant Value is selected.
  When creating input data for analysis "jinput", each condition should be specified as Constant Value on the assumption that conditions be modified by "jbdll.dll".
  Example: change the name "jplot" to "data1.plot".
- When conducting an analysis that uses the material database, etc., set the environmental variables of the JMAG installation directory or designate the function Set_DatabaseDirectory discussed later on.

  ▶▶ Reference
  For more information about the environmental variables, see P.408 "3-58. Setting Environment Variables".

  In the case of batch file "do_jam.bat", the environmental variable is set. If it is a machine installed with JMAG-Studio, it is already described in the registry, so the place of such registry is designated. Start up the module prepared by using library "jbdll.dll" from this batch file.

- The sequence flow from compile to calculation is as follows.

  i. **Create an executable module (exe. file) using library "jbdll.dll".**

  ii. **Write a command to run the executable module in the batch file.**

  If environmental variable setting is not carried out through a batch file, there is no need to create a batch file.

  iii. **Make up an input file for magnetic field analysis.**

  Modifiable variables are restricted in case of functions in the library are used. For example, in a case that electric potential setting is to be modified, it is necessary to select a constant value, instead of a point sequence, for setting the electric potential in supply voltage component on input data for magnetic field analysis.

  iv. **Execute the batch file.**

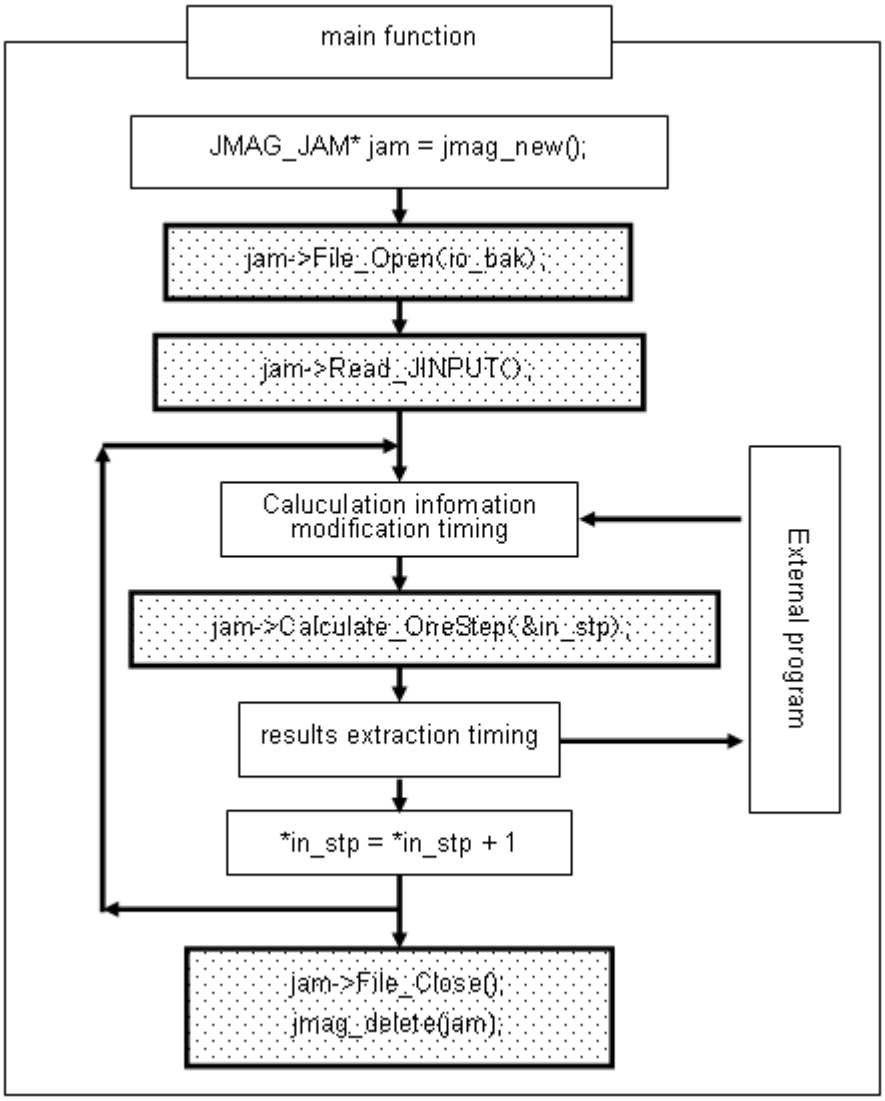
  When environmental variables are not specified by a batch file, execute the batch file in the usual way.

  After the calculation is completed, a file "jplot" is created as an analysis results file.

  v. **Modify the name "jplot" to "data.plot" and the like, and verify the results in JMAG-Studio.**

- The "JMAG_JAM" class is used to run calculation using this module. The class manages memory in which data required for calculations is stored, and functions contained in the class modify or extract data in the memory. When calculation is run using "JMAG_JAM" class, data in the memory is updated only when a calculation command is issued and received, instead of performing 2D analysis alone. In other words, modification of data in the memory is more appropriate than calculation. You therefore do not have to synchronize "JMAG_JAM" class with an upper-level external module that uses the class. In the external module process, you only have to issue calculation commands using functions of the "JMAG_JAM" class when you want to increment the JMAG analysis by one step.
- In this module, you need to call four functions in sequence after declaring the "JMAG_JAM" class as shown below. As functions are respective for startup processing and calculation at each calculation step, parameters for calculation such as electric current at each step can be modified from the external program. These are functions for processing at the start of calculation and calculation functions used at each step. In calculation information modification timing, functions designed to change calculation conditions can be used and in results extraction Timing, functions to obtain results can be used.

- Create "JMAG_JAM" with function "jmag_new", and always delete with "jmag_delete".



Execution flow of JMAG library

- You can re-run calculation with the original data by executing "jam_exe. Calculate_OneStep (&in_stp)".

3-57-1-1. Functions of JMAG_JAM Class

Functions of "JMAG_JAM" class are grouped into three types according to their properties.

- Starting and ending process:

  Executes a calculation and carries out the file reading process, including those functions inevitably used for calculating.
- Function for extracting calculation results:

  Obtains numerical values of results output at each calculation step from the memory region.
- Function for modifying calculation information parameters:

  Inputs values into the memory region to modify calculation information parameters before the start of each calculation step.

Important **Create "JMAG_JAM" with function "jmag_new", and always delete with "jmag_delete".**

The functions available are listed below:

■ **Functions for executing magnetic field analysis and load ing files:**

```
int File_Open(int io_bak);
        // Must call this function at the start of calculation
int File_Close();
        // Must call this function to quit calculation
int Read_JINPUT();
        // Function to load analysis input data
int Calculate_OneStep(int *in_stp);
        // Run only a specified calculation step
int Calculate_AllStep(int *in_stp);
        // Run all calculation steps
int Set_DatabaseDirectory(char* dir);
        // Function that designates material data base directory int
Set_SolverFileName(char* filename);
        // Function that designates solver data name
```

■ **Functions to extract calculation results:**

```
int Get_TotalStep(int *nstep);
        // Get the total step number of analysis
int Get_Torque(const int Group_Id,double *torq);
        // Get torque value (N m) of specified group
int Get_Force(const int Group_Id,double *force);
        // Get Electromagnetic Force (N) of specified group
int Get_Displacement(const int Group_Id, double *o_disp);
        // Get displacement (m, m, m, deg) of specified motion group
int Get_CircuitVoltage(const int Term_Id,double *volt);
        // Get electric potential (V) of specified terminal
int Get_CircuitCurrent(const int Line_Id,double *curr);
        // Get electric current (A) of specified component
```

■ **Functions to modify calculation data parameters:**

```
int Set_StepInTable_OutputControl(const int Point_Id,const int
Step);
        // Change step value of output control of each step.
int Set_OnOffINInTable_OutputControl(const int Point_Id,const
int Output_interval);
        // Change ON/OFF status on output control of each step.
int Set_DisplacementInConstant_Motion(const int Group_Id,const
double *i_disp);
        // Change specified displacement value for a specified motion.
int Set_VoltageInConstant_VoltageFunction(const int
VoltageFunction_Id,const double *Voltage);
        // Change constant voltage of " electric potential source components".
int Set_CurrentInConstant_CurrentFunction(const int
CurrentFunction_Id,const double Current);
        // Change constant current of "current source components".
int Set_TimeInConstant_Step(const double Time);
        // Change current time of a constant value setting in "step condition".
int Set_PreTimeInConstant_Step(const double PreTime);
        // Change previous time of a constant value setting in "step condition".
```

■ **Function for generation and extinction of JMAG_JAM:**

```
JMAG_JAM *jmag_new();
        // Function for creating JMAG_JAM.
void jmag_delete(JMAG_JAM *jam_exe);
        // Function for deleting JMAG_JAM.
```

3-57-1-2. Description of JMAG_JAM Functions

`0-1.Constructor ()`

Function:	Creates objects for data management in magnetic field analysis.
Argument:	None
Return value:	None
Description:	Creates and initializes objects necessary to magnetic field analysis.

`1-1.int File_Open(int _io_bak)`

Function:	Opens a file to be used for magnetic field analysis.
Argument:	When “in_bak” equals 0: Displays a calculation process on the standard output. When equal to 1: Outputs a calculation process to “jlpr2” and displays nothing on the standard output.
Return value:	None
Description:	This function must always be used. This function must be used once in performing magnetic field analysis. This function opens an input/output file for magnetic field analysis.

`1-2.int File_Close()`

Function:	Close the file used for magnetic analysis.
Argument:	None
Return value:	None
Description:	This function must always be used. This function must be used once in performing magnetic field analysis. This function closes the file used for magnetic field analysis.

1-3.int Read_JINPUT()

Function:	Loads an analysis input file (solver data file).
Argument:	None
Return value:	None
Description:	This function must always be used. Void File_Open () This function must be used once immediately after executing void File_Open () function This function loads analysis input data file (solver data file).

1-4.int Calculate_OneStep(int * in_stp)

Function:	Runs only the analysis of the set step.
Argument:	in_stp: Step number at which calculation is run
Return value:	1: Normal end 0: When calculation was not completed after the elapse of the number of total steps specified in a motion calculation with a distance specified.
Description:	Runs analysis with all analysis steps set in "in_stp". The value of in_stp should be in increments by 1 immediately after this function was called.

1-5.int Calculate_AllStep(int * in_stp)

Function:	Executes all calculation steps.
Argument:	in_stp: Start number of step where calculation starts.
Return value:	1: Normal end 0: When calculation was not completed after the elapse of the number of total steps specified in a motion calculation with a distance specified.
Description:	Runs all calculation step starting from the number specified by in_stp. Calculation data parameters cannot be modified midway, as calculation continues to run to the end step.

1-6.int Set_SolverFileName(char * dir)

Function:	Designates material data base directory.
Argument:	Designates JMAG installation directory.
Return value:	1
Description:	When there is no setting, conduct the calculation by assuming that there is a material data base directory in the directory that is being calculated.

1-7.int Set_SolverFileName(char * filename)

Function:	Designates solver data file name to be calculated.
Argument:	Designate the solver data file name.
Return value:	1
Description:	Place the solver data file in the work directory to be calculated, and designate only the solver data file name. Do not designate the bus.

2-1.int Get_TotalStep(int *nstep)

Function:	Obtains the total step number.
Argument:	None
Return value:	nstep: Total step number
Description:	Obtains the total step number that is described in the analysis input file (solver data file).

2-2.int Get_Torque(const int Group_Id,double *torq)

Function:	Obtains the torque value of the specified electromagnetic force and torque conditions group.
Argument:	Group_Id: The group ID of electromagnetic force and torque conditions
Return value:	torq: Torque (N·m) Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	Obtains the torque value of a specified electromagnetic force and torque conditions group. The torque obtained in the last calculation is acquired.

2-3.int Get_Force(const int Group_Id,double *force)

Function:	Obtains the electromagnetic force value of the specified electromagnetic force and torque conditions group.
Argument:	Group_Id: The group ID of electromagnetic force and torque conditions
Return value:	force [3]: Three components (x, y, z) of an electromagnetic force (N) Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	Obtains the electromagnetic force value of a specified electromagnetic force and torque conditions group. Finally the calculated results of electromagnetic force can be obtained.

2-4.int Get_Displacement(const int Group_Id, double *disp)

Function:	Obtains the displacement from the initial position of a specified "Motion" conditions group.
Argument:	Group_Id: The group ID of "Motion" conditions
Return value:	disp [4]: Displacement from an initial position X (m), Y (m), Z (m), and R (deg) Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	Obtains the displacement from the initial position of a specified "Motion" conditions group. Four components, X-direction, Y-direction, Z-direction and Rotational displacements, are taken in the form of an array. Displacements obtained in the last calculation are acquired.

2-5.int Get_CircuitCurrent(const int Line_Id,double *curr)

Function:	Obtains the electric current flowing through the component of a specified "Circuit" conditions.
Argument:	Term_Id: Terminal number
Return value:	volt [2]: Electric potential (A) (complex number) Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	The electric potential generating from the terminal of a specified "Circuit" conditions can be taken. The electric potential obtained in the last calculation is acquired.

```
2-6.int Get_CircuitCurrent(const int Line_Id,double *curr)
```

Function:	Obtains the electric current flowing through the component of a specified “Circuit” conditions.
Argument:	Line_Id: Component number
Return value:	curr [2]: Electric current value (A) (complex number) Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	The electric current flowing through the component of a specified “Circuit” conditions can be taken. The electric current obtained in the last calculation is acquired.

```
3-1.int Set_StepInTable_OutputControlconst int Point_Id, const
int Step);
```

Function:	Changes step value of point sequence for “Output Control of Each Step”.
Argument:	Point_Id: Point number in a point sequence Step: Step number to be modified.
Return value:	Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	Changes point sequence for “Output Control of Each Step”.

```
3-2.int Set_OnOffInTable_OutputControl(const int Point_Id,
const int Output_interval);
```

Function:	Changes output ON/OFF of point sequence for “Output Control of Each Step”.
Argument:	Point_Id: Point number in a point sequence. Output_Interval: Step interval to be output.
Return value:	Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	Changes point sequence for “Output Control of Each Step”.

```
3-3.int Set_DisplacementInConstant_Motion(const int Group_Id,
const double i_disp)
```

Function:	Changes displacement to the next step for a specified "Motion" conditions group.
Argument:	Group_Id: Group ID of "motion conditions". i_disp [4]: Distance from the position of the previous step.
Return value:	Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	If the specified "Motion" conditions group is set to Specify Constant Displacement, "Constant Displacement" parameters can be modified. Otherwise, this function cannot be used. A 4-element double type array should be used for the argument. From the top element, X-direction (m), Y-direction (m), Z-direction (m) and R-direction (deg) displacements.

```
3-4.int Set_VoltageInConstant_VoltageFunction(const int
VoltageFunction_Id,const double *Voltage)
```

Function:	Modifies the constant electric potential value of a specified "supply voltage" components.
Argument:	VoltageFunction_Id: Supply voltage component ID to be modified. Voltage [2]: Voltage to be set (V) (complex number).
Return value:	Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	If the specified "supply voltage" components are defined by specify constant electric potential, the electric potential can be modified. Other setting methods cannot modify the constant electric potential.

```
3-5.int Set_CurrentInConstant_CurrentFunction(const int
CurrentFunction_Id,const double *Current)
```

Function:	Modifies the constant current value of a specified "current source" components.
Argument:	CurrentFunction_Id: Current source component ID to be modified. Current [2]: Current to be set (V) (complex number).
Return value:	Return Value = 1: Value was acquired normally. Return Value = 0: Value was not acquired.
Description:	If the specified “current source” components are defined by specify constant current, the constant current value can be modified. Other setting methods cannot modify the constant current.

```
3-6.int Set_TimeInConstant_Step(const double Time);
```

Function:	Modifies the current time if “Step Function” conditions are defined by Set Constant Value.
Argument:	Time: Current time value to be modified (sec) or (Hz).
Return value:	1
Description:	Current time can be set.

```
3-7.int Set_PreTimeInConstant_Step(const double PreTime);
```

Function:	Modifies the previous time if “Step Function” conditions are defined by Set Constant Value.
Argument:	PreTime: Previous time value to be modified (sec) or (Hz).
Return value:	1
Description:	Previous time can be set. The previous time is needed for transient calculation. The same time value as current time should not be specified as the previous time.

■ Other Functions

```
4-1.JMAG_JAM *jmag_new();
```

Function:	Creates Class JMAG_JAM.
Argument:	None
Return value:	Object of JMAG_JAM.
Description:	Description:Creates the object of JMAG_JAM used in the calculation.

```
4-2.void jmag_delete(JMAG_JAM *jam_exe);
```

Function:	Deletes Class JMAG_JAM.
Argument:	Object of JMAG_JAM that does not use jam_exe.
Return value:	1
Description:	Releases the memory of JMAG_JAM that has already been used.

3-57-1-3. Example: "jmag_jam.h" File

In this file, input and output of "JMAG_JAM" class functions are described.

```
// JMAG_JAM.h
class JMAG_JAM
{
public:
  virtual int File_Open(int io_bak) = 0;
  virtual int File_Close() = 0;
  virtual int Set_DatabaseDirectory(char* dir) = 0;
  virtual int Set_SolverFileName(char* filename) = 0;
  virtual int Read_JINPUT() = 0;
  virtual int Calculate_OneStep(int *istep) = 0;
  virtual int Calculate_AllStep(int *istep_start) = 0;
  virtual int Get_TotalStep(int *nstep) = 0;
  virtual int Get_Torque(const int Group_Id,double *torq) = 0;
  virtual int Get_Force(const int Group_Id,double *force) = 0;
  virtual int Get_Displacement(const int Group_Id,double *disp)
= 0;
  virtual int Get_CircuitVoltage(const int Term_Id,double
*volt) = 0;
  virtual int Get_CircuitCurrent(const int Line_Id,double
*curr) = 0;
  virtual int Set_StepInTable_OutputControl(const int Point_Id,
const int Step) = 0;
  virtual int Set_OnOffInTable_OutputControl(const int
Point_Id, const int Output_interval) = 0;
  virtual int Set_DisplacementInConstant_Motion(const int
Group_Id, const double *i_disp) = 0;
  virtual int Set_VoltageInConstant_VoltageFunction(const int
VoltageFunction_Id, const double *Voltage) = 0;
  virtual int Set_CurrentInConstant_CurrentFunction(const int
CurrentFunction_Id, const double *Current) = 0;
  virtual int Set_TimeInConstant_Step(const double Time) = 0;
  virtual int Set_PreTimeInConstant_Step(const double PreTime)
= 0;

};

extern "C" _declspec(dllexport) JMAG_JAM *jmag_new();
extern "C" _declspec(dllexport) void jmag_delete(JMAG_JAM
*jam_exe);
```

3-57-1-4. Creating Executable Module

The executable module can be generated using the following procedures. An example of the main Function is shown in Sample 1.

1. **Start Visual Studio 2010.**
2. **Select [File] > [New] > [File] from the menu bar.**

   The [New Project] dialog box is displayed.
3. **Select [Win32 Console Application]. Enter “jam_run” as the project name, and then click [OK].**
4. **Click [Next] in the [Win32 Application Wizard] dialog box. Select the [Empty Project] check box, and then click [OK].**
5. **Select [Project] > [Properties], and then select [Configuration Properties] > [Linker] > [Input] > [Additional Dependencies] in the [Property Pages] dialog box. Add “jbdll.lib” to [Additional Dependencies].**
6. **Place “jbdll.dll” and “jmag_jam.h” under [Workspace Project].**
7. **Select [File] > [New] > [File], and then select [C++ File] in the [New File] dialog box. Click [Open].**

   The [Source1.cpp] tab is displayed.
8. **Right-click the [Source1.cpp] tab, and then select [Save Source1.app As] from the shortcut menu.**
9. **Enter “main.cpp” as the file name, and then save the file under [Workspace Project].**
   Drag “main.cpp” to [Source File] under [Solution Explorer].

   A sample module ‘main.cpp‘can also be added to [Source File] under [Solution Explorer].
10. **Put any code in ‘main.cpp’.**

    “jam_run” is created as a result. A sample module is located in the installation directory of JMAG-Studio or JMAG-Designer “\solver\mod\mag\usedll”.
11. **Copy DLL that is required for executing the program, and paste it under [Workspace Project].**

    For example, in the case of Win32, filehandler_32_05.dll, UnitLibrary_32_05.dll, xerces-c_1_6_0_32_05.dll, icuuc20.dll, icudt20.dll, RT_DLL_32_05.dll etc.

3-57-1-5. Debugging

For reference, the following shows the method for debugging.

1. **Create the Debug version of user-defined DLL.**

   If using the Simulink, create the Debug version of S-function DLL.

2. **Enter command at command prompt to start Visual C++.**

   Enter the following command.

   ```
   msdev (The name of user-defined DLL)
   ```

   If using the Simulink, enter the name of S-function DLL.

3. **Select [Project] > [Configure].**

4. **Select Debug.**

5. **Specify the starting module of the program that was generated in the executable file in the debug session.**

   If Release 11 was installed in the C-drive, it is displayed as follows.
   "C:\matlabR11\bin***.exe" The name of the generated program appears in ***.

6. **Click [OK].**

7. **Select [Build] > [Debug] > [Run].**

8. **Click [OK] when the message "no debug information" is displayed in the generated program.**

9. **Execute debug as the generated program starts.**

3-57-1-6. Sample Programs: “main.cpp” File

After running JMAG, the total step number is read, and calculation is performed for the total steps. Rotational displacement is directly specified each time prior to starting each step calculation. The displacement is multiplied by two for each step.

```
#include <iostream>
#include <fstream>
#include “jmag_jam.h”
int main(int argc,char **argv)
{
int in_stp = 1; int nstep;
double i_disp[4]; double o_disp[4];
double time,pretime;
double torque = 0.0; double voltage[2]; double current[2];
     i_disp[0] = 0.0;
     i_disp[1] = 0.0;
     i_disp[2] = 0.0;
     i_disp[3] = -2.0;
     JMAG_JAM* jam = jmag_new();
     jam->Set_SolverFileName("motor_dll");
     jam ->File_Open{1};
                              // Start calculation
jam ->Read_JINPUT();
                              // Read solver data file
     jam ->Get_TotalStep(&nstep);
                              // Read the total step number
    pretime = 0.0;
      for(int k=in_stp;k<=nstep;k++){
         time = (k-1) * 1.0;
                              // Calculate every 1sec
         jam->Set_TimeInConstant_Step(time);
         jam->Set_PreTimeInConstant_Step(pretime);
jam->Set_DisplacementInConstant_Motion(1,i_disp);
                              // Specify rotation angle
jam->Calculate_OneStep(&k);
                              // Run the k-th step of analysis
i_disp[3] = i_disp[3]*2;
                              // Multipy displacement by two
jam->Get_Displacement(1, o_disp);
                              // Get current displacement, x, y, z, r
jam->Get_Torque(1,&torque);
                              // Get torque (Nm)
jam->Get_CircuitVoltage(2,voltage);
                              // Get electric potential (V) at No. 2 terminal
```

```
jam->Get_CircuitCurrent(2,current);
                                    // Get electric current (A) at No. 2 component
          pretime = time;
         }
         jam->File_Close();
                                    // Terminate JMAG(DP) calculation
       jmag_delete(jam);
return 1;
}
```

3-58. Setting Environment Variables

To operate a running module using the material database, it is necessary to set environmental variables for JMAG. There are two ways to do this. One is to use a batch file in which commands for setting environment variables and for invoking the executable module are described. The other is to register the environment variables in the computer. For either method, JMAG-Studio must be installed in the machine.

3-58-1. Using a Batch File to Set Environmental Variables

The following is an example of a batch file named “do_dpl.bat”. The environment variable “RegPath” is set in the file. The file name dpl_dun.exe is the executable module, which is run by this batch file. The value given to “RegPath” must be in the location in which JMAG-Studio data is stored in the Registry, instead of the installation directory.

■ **Sample batch file, “do_dpl.bat”**

```
------------------------------------------------------------
@echo off
set RegPath=Software\JRI\JMAG-Studio10.0
.\jamrun.exe
------------------------------------------------------------
```

3-58-2. Setting the Environmental Variables on Your Computer (For Windows XP)

When a registry is made for the computer environmental variables, it is not necessary to conduct setting of the environmental variables each time at the start up of the execution module. As a result, it is not necessary to prepare batch files. Execute the execution module with the command prompt.

1. **Double-click the [My computer] button.**
2. **Double-click the [Control Panel] button.**
3. **Double-click the [System] button.**
4. **Click the [Detailed Setting] tab and the [Environment Variables] button.**
5. **Click [New] in the [System Environment Variable] group box and open the [New System Variables] dialog box.**
6. **Input "RegPath" to [Variable Name] and "Software\JRI\JMAG-Studio10" to [Variable Value].**
7. **Click [OK] and close all dialog boxes.**

3-58-3. Checking the Registry (For Windows XP)

1. **Select [Start] - [Run...].**
2. **Type "regedt32" and then click [OK].**
3. **Double-click [Software] in HKEY CURRENT USER.**
4. **Double-click [JRI] to confirm where JMAG-Studio data is located.**

3-59. Output Items of Magnetic Field Analysis

3-59-1. View Type and Display Method

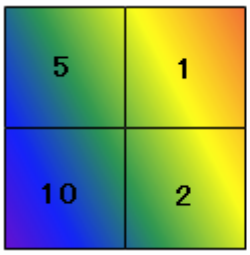
View types are categorized according to the component as follows.

■ Step:

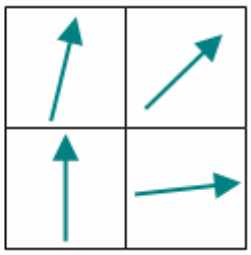
Displays the result for one step (or multiple steps).

■ Distribution quantity:

- Scalar:
  Displays a colored image
- Vector:
  Displays an arrow image
- Three components:
  Displays the scalar in the x, y, and z directions



Scalar



Vector

Drawing type		Vector	Contour	Flux line	Section graph	Table	History graph
Step		-	-	-	-	*	*
Distribution quantity	Scalar	-	*	-	*	*	*
	Vector	*	*	* In the case of solid	*	*	*
	Three components (x,y,z)	-	*	-	*	*	*

3-59-2. Output Items

Magnetic field analysis modules (ST, TR, FQ, and DP) can display the following items as analysis results:

MEMO Each of the export items are displayed in units that are applied to an analysis model in JMAG-Designer.

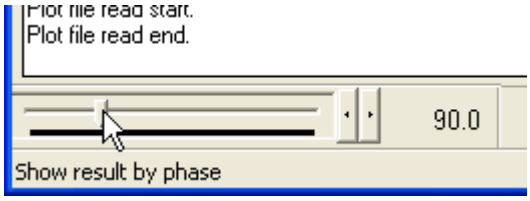
Output item	Unit	Type	Object	Note
Magnetic field	A/m	vector	element	
Magnetization	A/m	vector	element	
Magnetic flux density	T	vector	element	
Magnetic flux	Wb	step	-	
Flux of FEM coil	Wb	step	-	For the calculation formula, see MEMO.
Magnetic flux in Current condition	Wb	step	-	For the calculation formula, see MEMO.
Magnetic flux lines	None	*Refer to the Note.	element (shell)	Contour type of display setting does not apply. It will always display in linear display. Only for 2D analysis.
Vector potential	Wb/m	vector	node	Export as ST (2D,AX), FQ (2D,AX), DP (2D,AX). For 2D, vector potential is exported. For AX, vector potential times radius (distance from the Y-axis) is output.
Electric field	V/m	vector	element	This will be output in case relative permittivity is specified in the FQ.
Current density	A/m^2	vector	element	
Surface current density	A/m	vector	element (shell)	This will be output when using SIBC function in FQ (3D).
Current	A	step		

Output item	Unit	Type	Object	Note
Electric potential	V	step	-	
Electric power	W	step	-	
Joule loss density	W/m^3	scalar	element	
Surface Joule loss density	W/m^2	scalar	element (shell)	This will be output when using SIBC function in FQ (3D).
Joule loss	W	step	-	Output for each material ID and summation
Hysteresis loss density	W/m^3	scalar	element	
Hysteresis loss	W	step	-	Output for each material ID and summation
Torque	N·m	step	-	Output actual value for FQ.
Electromagnetic force	N	step	-	Output actual value for FQ.
Nodal force	N	vector	node	The nodal force is outputted for the 0-order component, and 2-order component in FQ analysis.
Nodal Force (magnetostriction)	N	step	-	Frequencies in a period for specified step is outputted for FQ.
Surface force density	N/m^2	vector	element face	Output actual value for FQ.
Lorentz force density	N/m^3	vector	element	The nodal force is outputted for the 0-order component, and 2-order component in FQ analysis
Lorentz force	N	step	-	Output actual value for FQ.
Displacement	m	vector	node	The amount of displacement from the previous step at each node is obtained.

Output item	Unit	Type	Object	Note
Speed	Rotational motion: rpm Translational motion: m/sec	step	-	For more information see, "3-17-2-4. Displacement" on page 186.
Total displacement	Rotational motion: deg Translational motion: m	step	-	This will be output if motion condition is set in TR, DP.
Stored energy	J	step	-	Output for each material ID and summation
Relative permeability	N/A	contour	element	
Differential permeability	N/A	contour	element	
Demagnetizing ratio	%	scalar	element	For the calculation formula, see MEMO mentioned later.
Permeance coefficient	none	scalar	element	
Initial magnetization for permanent magnet	A/m	vector	element	
Winding inductance	H	step	-	Only for Transformer analysis of JMAG-Designer.
Winding DC resistance	ohm	step	-	Only for Transformer analysis of JMAG-Designer.
Flux linkage in coil	Wb	step	-	Only for Transformer analysis of JMAG-Designer.

MEMO If an iron loss condition is set, the results of the iron loss analysis can also be obtained. For information about the output items of iron loss analysis, see "Displaying Results (Output Items of Iron Loss Analysis)" on page 904.

MEMO Result display of real and imaginary parts (frequency response analysis (FQ))
To display the results of a real part, set the phase to 0 using the slider.
To display the results of an imaginary part, set the phase to 90 using the slider.



Slider of JMAG-Studio

MEMO Formula for calculating the demagnetizing ratio
The calculation formula of the demagnetization ratio is as follows.

$$\text{Demagnetizing ratio (\%)} = 100 \times \left(1 - \frac{B2}{B1}\right)$$

B1: Residual magnetic flux density of the specified step. Specify the step number in the [Specify Step] dialog box for JMAG-Studio. Select [Results] > [Demagnetizing Ratio] > [Setting] to display the [Specify Step] dialog box. Specify the step numbers when creating/editing the contour plot or the section graph in JMAG-Designer.
B2: Residual magnetic flux density of the step that is displayed.

MEMO How to calculate the volume average of magnetic flux density
1) Select [Mesh] > [Element] > [Quality] in the graphics window in JMAG-Studio.
2) Output element volume to a file.
3) Indicate values of magnetic flux density at the points to be required for the result window by using a table.
4) Output to a file the indication contents of Procedure (3).
5) Perform calculations using a commercial calculation tool.

MEMO Formula for calculating the magnetic flux in the FEM coil and the magnetic flux with the current condition

$$\Phi = \Phi_1 \times N$$

Φ_1: Magnetic flux per turn (Wb)

N: Number of turns

MEMO Relation between magnetic flux density B (T), magnetic field H (A/m), and magnetization M (A/m) in the following formula.

$$B = \mu_0(H + M) \quad (\mu_0: \text{permeability in vacuum})$$

In JMAG, a magnetic field is calculated from magnetization data calculated from magnetic flux density and magnetizing properties that have been set.

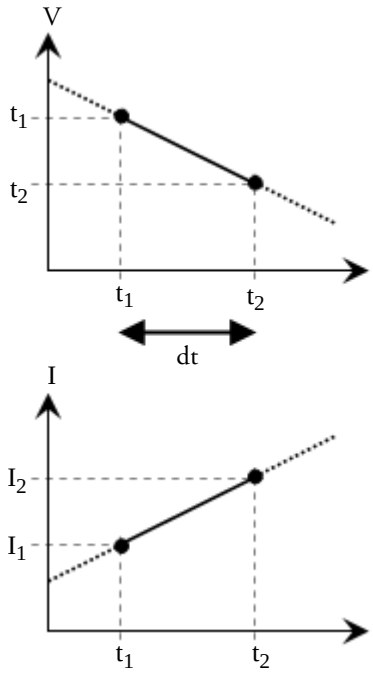
MEMO Magnetic Flux of FEM coil for 2D Analysis Results
- The magnetic flux of the FEM coil is calculated using the value specified in [Stack Length] if the [Convert to Full Model] check box is selected and the [Height of FEM Coil/Conductor Differs from Stack Length] check box is not selected.
- The value of the stack length for the magnetic flux is calculated as 1 m if both are converted to full model and the height of the FEM coil/conductor is from the same as the stack length. [Height of FEM Coil/Conductor Differs from Stack Length] check box is not selected.

MEMO Definition formula for electric power (transient response analysis (TR, DP)).
When current and voltage are expressed respectively with (t_1, V_1, I_1) and (t_2, V_2, I_2) within an interval (t_1, t_2), the electric power W is:

$$W(t_2) = \frac{1}{\Delta t}\int_{t_1}^{t_2} V(t) \cdot I(t)dt$$

$$\Delta t = t_2 - t_1$$

This W is handled as the value for t_2.The value of W at the initial time is 0.0.

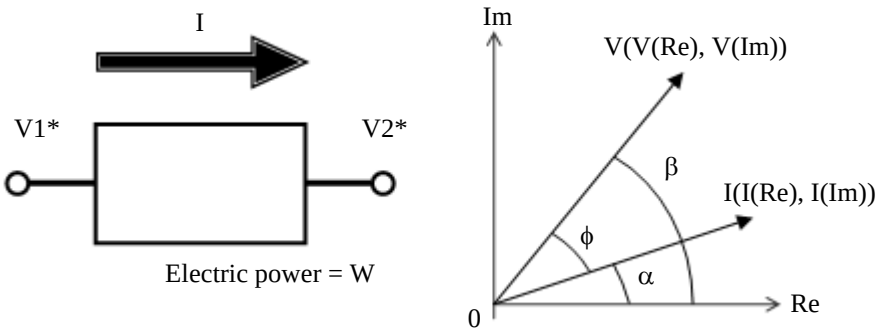


MEMO Definition formula for electric power (frequency response analysis (FQ).

Active power: $W = Veff*leff*\cos\phi = VI\cos\phi/2$

Reactive power: $W = Veff*leff*\sin\phi = VI\sin\phi/2$

Apparent power: $W = Veff*leff = VI/2$



The terminal voltage across the first and second terminal of the component is V_1^* and V_2^*, respectively.
The current amplitude of the component is *I* and its effective value is *Ieff*.
The electric potential difference is *V* and its effective value is *Veff*.

Electric potential difference $V^* = V_2^* - V_1^*$

MEMO Displaying permeance coefficient in the magnetic field analysis

(1) In the result display window, select the magnet element.
(2) On the menu bar, select [View] > [Permeance Coefficient] > [Contour] (or [MinMax-History).
Selecting [MinMaxHistory] displays the history graph of the maximum and minimum permeance coefficient within the selected element.

MEMO Permeance coefficient

The permeance coefficient can be expressed by the following equation,

$$Pc = -Bd/(\mu_0 Hd)$$

where Bd and Hd represent flux density and magnetization component of the magnetic field, respectively.
Permeance coefficient is displayed in the second and third quadrants of the magnetizing properties. When the moving point is in the first quadrant, the maximum value appears in the second quadrant. In that case, a message containing the material ID will be output along with the solver report. When the magnetic field is small, the permeance coefficient becomes large. So, when viewing it on the model, adjust the maximum level of the color range to an appropriate level by selecting [Options] > [Result Display Preferences].

MEMO Hysteresis loss (negative value)

When the complex relative permeability is specified for the magnetizing properties in the frequency response analysis (FQ), the hysteresis loss is obtained. However, if the negative value is specified for the imaginary part, the hysteresis loss is exported in the negative value.

Chapter 4. Function of Thermal Analysis

This chapter describes in detail how to set the material properties and analysis conditions used in thermal analysis.

▶▶ Reference

For more information about the procedures, see the JMAG-Studio User's Manual or the JMAG-Designer User's Manual.

4-1. Overview of Functions

4-1-1. HT (Thermal Analysis Module)

- HT module uses the finite element method (FEM). Nodal temperature can be prescribed or set as unknown.
- HT module can perform thermal analysis for stationary state or transient.
  - Stationary state implies that the temperature distribution is independent of time.
  - Transient implies that the temperature distribution is time dependent.
- Setting motion condition enables you to represent the motion of a heated object.
- HT module can be coupled with magnetic analysis (FQ or TR module).
- HT module can be coupled with electromagnetic wave frequency response analysis (WV).

Important **Only 3D models can be used for thermal analysis.**

▶▶ Reference

For more information, see P.769 "Chapter 10. Coupled Analysis".

4-2. Conditions of Thermal Analysis (JMAG-Designer)

The following conditions are available for the thermal analysis in JMAG-Designer.

Analysis Conditions	Module
	HT
Material	
Material properties	⊙
Boundary	
Periodic boundary (Rotation/Translation)	O
Natural boundary	O
Constant temperature boundary	O
Equivalent temperature boundary	O
Heat transfer boundary	O
Radiation heat transfer boundary	O
Heat flux boundary	O
Initial temperature	
Initial temperature	O
Heat source	
Heat source	O
Motion	
Motion (Rotation/Translation)	O
Output	
Average temperature	O
Heat flow	O

Analysis Conditions	Module
	HT
Modeling	
Contact thermal resistance	O
Link	
Partial model	O
Circuit	
Thermal circuit	O
Study properties	
Step control	⊙
ICCG	O
Nonlinear calculation	O
Coupled analysis	O
Restart	O
Output control	O
User subroutine	O

⊙: Required O: Optional X: Not available

4-3. Conditions of Thermal Analysis (JMAG-Studio)

The following conditions are available for the thermal analysis in JMAG-Studio.

Analysis Conditions	Module HT
	Studio
Material properties	⊙
Motion	O
Heat flux boundary	O
Radiation heat transfer boundary	O
Temperature boundary	O
Constant temperature	O
Periodic boundary	O
Heat transfer boundary	O
Initial temperature	O
Heat source	O
Calculate heat flow	O
Contact thermal resistance	O
Equivalent thermal circuit	O
Calculate average temperature	O
Partial model	O
Time step	⊙
Analysis control	⊙
ICCG	O
Nonlinear calculation	O
Output control	O
User subroutine	O

⊙: Required O: Optional X: Not available

4-4. Material Property for Each Part

4-4-1. Endotherm Coefficient and Equilibrium Temperature

- Specifies the settings for endotherm coefficient and equilibrium temperature when calculating endotherm results on materials.
- The heat transfer effect is expressed by the following heat conduction equation.

$$\rho C\frac{\partial T}{\partial t}-\vec{\nabla}\cdot(k\vec{\nabla}T)-Q=0$$

ρ : Density of material object, C : Specific heat, T :Temperature,

t : Time, k : Thermal conductivity, Q : Quantity of heat

The quantity of heat including endotherm Q' when there is endothermic coefficient ϕ and equilibrium temperature T_{eq} is expressed by the following when using the actual quantity of heat Q .

$$Q'=Q-\phi(T-T_{eq})$$

MEMO The endotherm is used to reduce quantity of heat.
For example, when a low temperature liquid flows through a pipe inside the model, the temperature of the object around the pipe is decreased because of heat transfer. This function can express this effect without modeling the passage area.

4-4-2. Output Control (JMAG-Studio)

- Only materials for which [Output] is selected in the [Post Output] group box is displayed in the result window.
- Only settings related to the materials for which [Output] is selected are exported to the plot file.
- [Output] is selected by default.

4-5. Periodic Boundary (Rotation/Translation)

▶▶ Reference

- For more information, see "Periodic Boundary" in Chapter 10 Setting Conditions of Thermal Analysis in the JMAG-Studio User's Manual.
- For more information, see "Rotation Periodic Boundary" and "Translation Periodic Boundary" in Chapter 31 "Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-5-1. Function

When analysis model has axis of symmetry then employ the periodic boundary condition to reduce the analyzed region of the analysis model.

■ Related items

- N/A

■ Settings

- 1st periodic boundary face

  MEMO The 1st periodic boundary face is the first face that is counterclockwise from the two periodic boundary faces of the rotational model.

  Specifying the rotational/translation periodic boundary face

  Rotational periodic boundary
  - A point on rotation axis
  - Direction of rotation axis
  - Angle of rotation
  - Specifying periodic/antiperiodic boundary

  Translation periodic boundary
  - Direction of translation periodic
  - Periodic distance
  - Specifying periodic/antiperiodic boundary

■ Output items

- N/A

4-5-2. Important Notes Regarding the Use of This Function

- Select nodes on the first periodic boundary surface in the analysis model.
- The first periodic boundary is defined as the surface having the smaller angle on the vector boundary surface at both ends of the model with symmetry relative to the rotation axis. The second periodic boundary is defined as the surface having the larger angle.
- Nodes on the second periodic boundary surface must have corresponding nodes on the first periodic boundary surface. Because JMAG-Studio automatically determines the nodes on the second periodic boundary surface.
- Faces on which the boundary condition is not set are treated as having a heat insulation (heat flux is zero).

4-5-2-1. Periodic Boundary/Anti-periodic Boundary

- In the case of periodic boundary condition, when a model is rotated by the angle of rotation, size and direction of a certain physical quantities (e.g. temperature distribution) are unchanged before and after the rotation (see Fig.1).

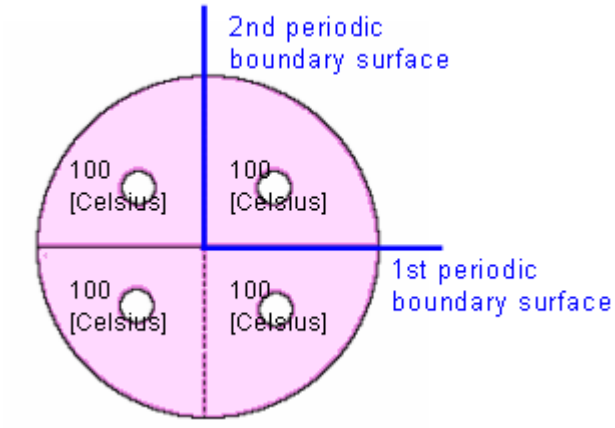


Fig.1 Periodic boundary

- In the case of Anti-periodic boundary condition, when a model is rotated by the angle of rotation, size of a certain physical quantities (e.g. magnetic flux density) is unchanged and its direction is opposite before and after the rotation (see Fig. 2).

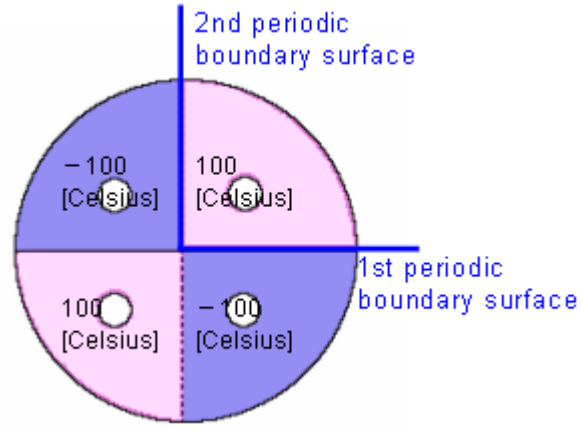


Fig.2 Anti-periodic boundary

4-5-2-2. Rotational Periodic Boundary

- "Point on rotation axis" means the coordinate for central section of the model, e.g., the coordinate is (0,0,0) for a model which is periodic around Z-axis centering the origin.
- "Direction of rotation axis" means a vector passing through center of rotation, e.g., if an axial vector is set to Z-axis, the axial vector is (0,0,1).
- "Periodic angle" means the angle of the center of the analysis model. For a 1/4 model the periodic angle is 90deg.

4-5-2-3. Translational Periodic Boundary

- "Translation direction" is the vector representing the direction of translation. For example, if the boundary has a translational periodicity in the X-axis direction, the direction is (1, 0, 0).
- For the translational periodic boundary, enter the periodic distance instead of the periodic angle. For example, if it has a periodicity every 30 cm, the periodic distance is 0.3 m.

4-6. Natural Boundary (JMAG-Designer)

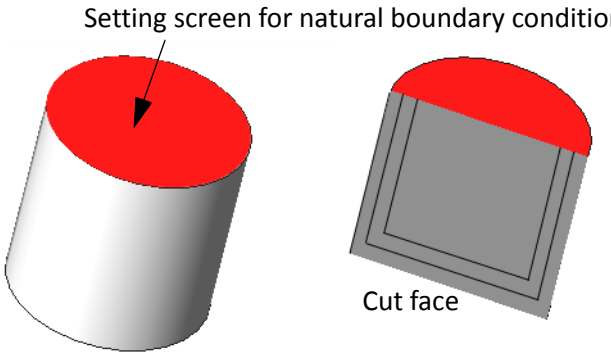
▶▶ Reference

For more information, see “Natural Boundary” in Chapter 31 “Study Properties and Conditions of Thermal Analysis” in the JMAG-Designer User's manual.

4-6-1. Function

The natural boundary condition is set to the analysis model. The face this condition is specified does not transfer heat.

MEMO The natural boundary condition is used to generate mesh simulating the skin effect accounting for the model cross-section.



Thin layered mesh modeling the skin effect generated on every face but the face the natural boundary condition is set.

■ Related items

- Thin layered mesh for modeling the skin effect (see JMAG-Designer User's Manual)

■ Settings

- Natural boundary face

■ Output items

- N/A

4-7. Constant Temperature Boundary

▶▶ Reference

- For more information, see "JMAG-Studio User's Manual," "Chapter 10. Setting Conditions of Thermal Analysis," "Temperature Boundary"
- For more information, see "JMAG-Designer User's Manual," "Chapter 31 Study Properties and Conditions of Thermal Analysis," "Constant Temperature Boundary."

4-7-1. Function

Sets the temperature for parts and faces that are specified.

MEMO The constant temperature boundary and the equivalent temperature boundary are set in the temperature boundary condition in JMAG-Studio. Specify the temperature by selecting [Fixed Temperature] in the temperature boundary condition when setting the constant temperature boundary.

■ Related items

- N/A

■ Setting

- Target (part, face)
- Temperature

■ Output item

- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

4-7-2. Important Notes Regarding This Function

- This condition is used to specify a constant temperature for a target. Specify the equivalent temperature boundary condition when setting a equivalent temperature (temperature is not fixed) for a target.

4-8. Equivalent Temperature Boundary

▶▶ Reference

- For more information, see “JMAG-Studio User's Manual,” “Chapter 10. Setting Conditions of Thermal Analysis,” “Temperature Boundary”
- For more information, see “JMAG-Designer User's Manual,” “Chapter 31 Study Properties and Conditions of Thermal Analysis,” “Equivalent Temperature Boundary.”

4-8-1. Function

This function constantly equalizes temperature (change in temperature) for specified parts and faces and connecting specified parts and faces to terminals of equivalent circuits.

MEMO The constant temperature boundary and the equivalent temperature boundary are set in the temperature boundary condition in JMAG-Studio. Specify the equivalent temperature boundary by selecting the [Constant Temperature] in the temperature boundary condition.

■ Related items

- Heat flux boundary
- Heat transfer boundary
- Radiation heat transfer boundary
- Thermal circuit

■ Setting

- Target (part, face)
- Use in thermal circuit

■ Output items

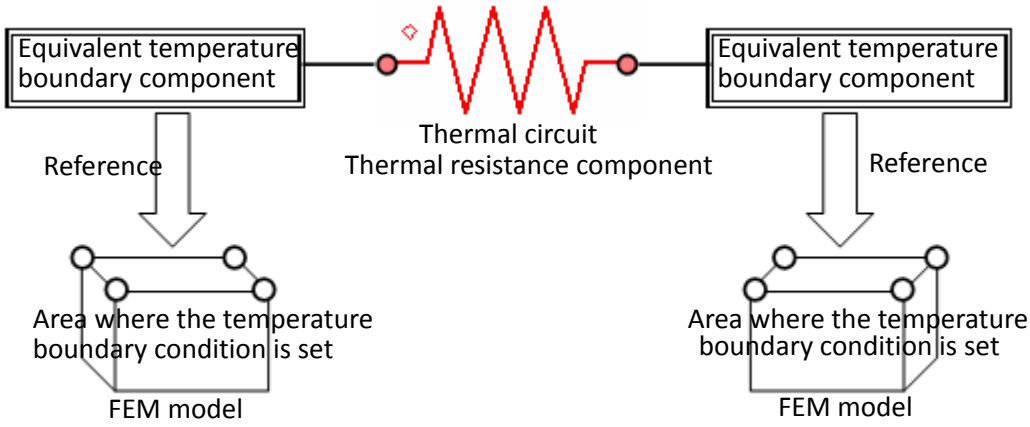
- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

4-8-2. Important Notes Regarding This Function

- This condition is used to set an equivalent temperature for a target (temperature is not fixed). Specify the constant temperature boundary condition when setting a constant temperature for a target.
- Specify the equivalent temperature boundary condition when the heat transfer boundary condition is not used to connect a thermal circuit.
- The temperature distribution specified by the initial temperature condition is exported for the first step regardless of steady state or transient state analysis.
- Multiple equivalent temperature boundary conditions cannot be set for the same node.
- Faces where the temperature boundary condition is not set are treated as having a heat insulation (heat flux is zero).

4-8-2-1. Applying to the Thermal Circuit

- An equivalent temperature boundary component needs to be connected to the terminal of the circuit to reference the equivalent temperature boundary condition from the circuit. With this setting, the temperature of the terminal connected to the equivalent temperature boundary component is equivalent to the temperature of the areas with the equivalent temperature boundary condition set.



Connection between the FEM models and thermal circuit using the temperature boundary condition

- In JMAG-Studio, if the [Apply to Equivalent Circuit Condition] radio button is selected for the temperature boundary condition, the temperature cannot be set.

4-9. Heat Transfer Boundary

▶▶ Reference

- For more information, see “Heat Transfer Boundary” in Chapter 10 Setting Conditions of Thermal Analysis in the JMAG-Studio User's Manual.
- For more information, see “Heat Transfer Boundary” in Chapter 31 “Study Properties and Conditions of Thermal Analysis” in the JMAG-Designer User's Manual.

4-9-1. Function

Defines heat transfer coefficient and reference temperature on material surfaces. The reference temperature under this condition means the ambient temperature around the analysis model. In addition, it is possible to connect the objects set in this condition to the terminals of the equivalent circuit.

■ Related items

- Radiation heat transfer boundary
- Equivalent temperature boundary
- Thermal circuit
- User subroutine: *usrhtc*

■ Targets

- Element face group
- Solid face group

■ Settings

- Reference temperature
  - Constant
  - Use the temperature specified for a thermal circuit
- Heat transfer coefficient
  - Constant value
  - Specify by point sequence
  - Use user subroutine

■ Output items

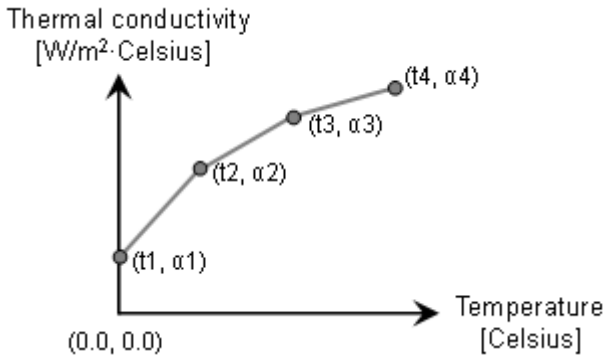
- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

4-9-2. Important Notes Regarding the Use of This Function

- If model symmetry is employed in the analysis then the heat transfer boundary condition cannot be prescribed on the planes of symmetry.
- Multiple heat transfer boundaries can be set. Regarding the setting conditions, reference temperatures from other conditions can be set.
- If specifying the heat transfer coefficient using point sequence, two or more points should be determined as shown below.

  Example) When the thermal conductivity is dependent on temperature, the point sequence should be set as show below.

Temperature	Thermal Conductivity
t1	1
t2	2
t3	3
t4	4

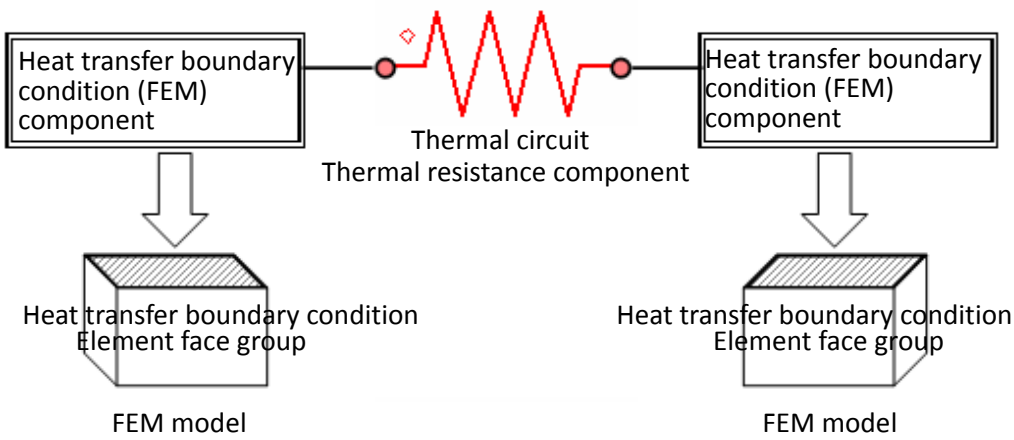


Point sequence
when thermal conductivity is dependent on temperature

- The thermal conductivity above can be changed by the user subroutine *usrhtc* (475 page).
- Faces on which the boundary condition is not set are treated as having a heat insulation (heat flux is zero).

4-9-2-1. Applying to the Thermal Circuit

- To reference the heat transfer boundary condition from the circuit, connect the heat transfer boundary component to the terminal in the circuit. With this setting, the temperature of the terminal connected to the heat transfer boundary component is used for the temperature of the element group with the heat transfer boundary condition set.



Connection between the FEM models and equivalent circuit using the heat transfer boundary condition

- In JMAG-Studio, if the [Apply to Equivalent Circuit Condition] radio button is selected for the heat transfer boundary condition, the reference temperature cannot be set.

4-10. Radiation Heat Transfer Boundary

▶▶ Reference

- For more information, see "Radiation Heat Transfer Boundary" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Radiation Heat Transfer Boundary" in Chapter 31 "Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-10-1. Function

Radiation heat transfer boundary condition enables the exchange of radiant energy between object surface and surrounding environment.

For example, heat emitted as electromagnetic waves from a face can be obsorbed to surrounding faces. Such as the amount of radiation from a face including the temperature from surrounding faces.

■ Related items

- Equivalent temperature boundary
- Heat transfer boundary
- User subroutine: *usrhrd*

■ Settings

- Radiation heat transfer boundary face
- Radiation coefficient (gray coefficient)
  - Constant
  - Point sequence
  - Using user subroutine *usrhrd*
- Reference temperature

■ Output items

- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

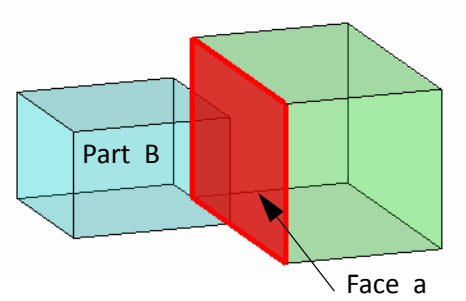
4-10-2. Important Notes Regarding the Use of This Function

- The heat flux can be obtained using the following formula.

  $$Q = \varepsilon\sigma F(T^4 - T_0^4)$$

  Q: heat flux, ε: radiation coefficient, σ: Boltzmann constant, F: configuration factor, T_0: reference temperature

  If the radiation energy is transferred between the surfaces of objects, T_0: is the surface temperature of the diagonal object.
- Faces that are not set with the boundary condition are used as a insulation (heat flux is zero).
- Specify a radiation heat transfer boundary condition for faces exchanging radiation energy.
  The transferring of radiation heat is run between faces set with the radiation heat transfer boundary condition. Radiation heat cannot be emitted or absorbed on faces that do not have the radiation heat transfer boundary condition set.
- Specify the radiation heat transfer boundary condition in the outer planes of the model.
  If it is determined that the inner planes and not the outer planes are set with the radiation heat transfer condition, an error occurs during an analysis. This happens when the face that is being set with the radiation heat transfer condition which is 'a,' is connected to part 'B.' This also applies if the [Include This Material to Analysis] flag for the connected part 'B 'is not selected, or if the material for the connected part 'B' is air.



Error due to settings for the radiation heat transfer boundary condition

The part area where there are no connections can be set for the radiation heat transfer boundary condition by setting [Imprint for parts] in the geometry editor. This can be specified if the area that is connected is partial as shown in the figure above.

▶▶ Reference

For more information about [Imprint for Parts] in the geometry editor, see "JMAG-Designer User's Manual."

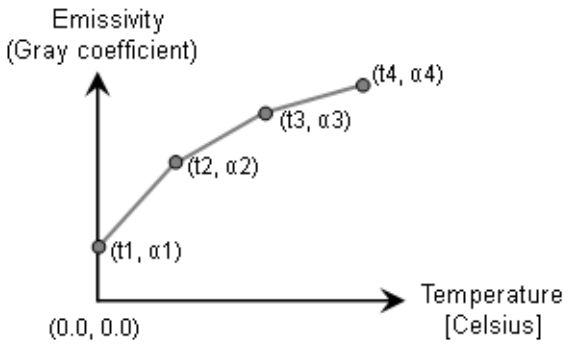
- This condition cannot be set on the cross-section of a partial model.

- The radiation heat is used regardless of difference in conditions, when multiple radiation heat transfer boundary conditions are set. For example, radiation heat emitted from a face set with radiation heat transfer boundary condition 'A,' can also be used for a face set with radiation heat transfer boundary condition 'B.'
- If a partial model is used in the analysis, only the radiation energy transferring on the surface of the partial model is taken into account.
- The radiation coefficient (grey coefficient) of the surface of an object and the reference temperature of the surrounding area needs to be set.
- The temperature of the surrounding area is referenced, if the radiation energy is transferred between an object and the surrounding area.
- To specify the radiation coefficient using a point sequence, specify two or more points. The following is an example.

■ Example)

If the radiation coefficient (gray coefficient) is dependent on temperature, the point sequence should be set as show below.

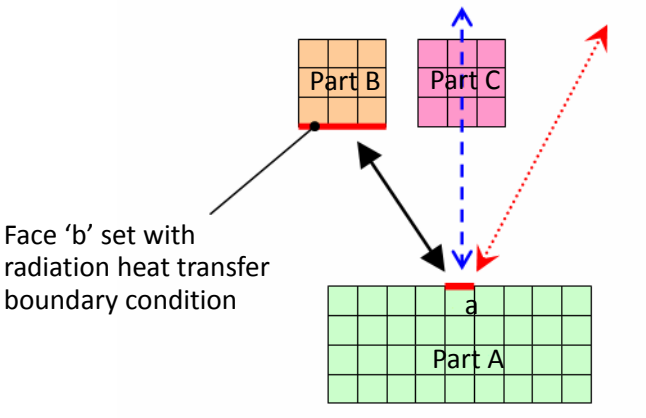
Temperature	Radiation coefficient (gray coefficient)
t1	1
t2	2
t3	3
t4	4



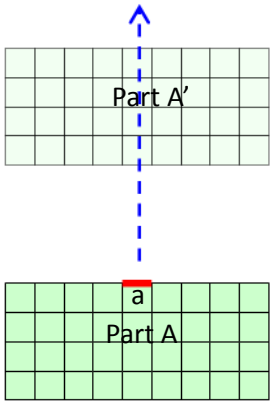
Point sequence when radiation coefficient (gray coefficient) is dependent on temperature

- The radiation coefficient α above can be changed by the user subroutine *usrhrd* (476 page).

- Note that the calculation for the radiation heat effect may differ based on the division method or condition setting of a model.
  For example, the following is when the radiation heat is from element face 'a.'
  The temperature on 'part B' is used for calculating radiation heat effect in the solid arrow direction. This occurs when the radiation heat transfer boundary condition set on face 'b' exists.
  Part 'C' exits in the direction of the dashed arrow, but the radiation heat transfer boundary condition is not set. The input value in [Reference Temperature] is used when calculating the direction for a radiation heat effect.
  A part does not exist in the dotted arrow direction. The direction for the radiation heat effect is also calculated by using the input value set in [Reference Temperature].



  The input value in [Reference Temperature] is used when calculating the radiation heat effect, when the radiation heat transfer boundary condition and the periodic boundary condition are both set on element face 'a.'



4-11. Heat Flux Boundary

▶▶ Reference

- For more information, see "Heat Flux Boundary" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Heat Flux Boundary" in Chapter 31 "Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-11-1. Function

Defines heat flux entering into a specified boundary surface.

■ Related items

- Equivalent temperature boundary

■ Settings

- Heat flux boundary face
- Heat flux

■ Output items

- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

4-11-2. Important Notes Regarding the Use of This Function

- If entering a positive value in the [Heat Flux] text box, the heat flux is treated as an inflow, and if entering a negative value, it is treated as an outflow.
- This condition can prescribed on the boundary surface (outermost surface) of the analysis model.
- Always employ this condition together with temperature boundary. If this condition is solely used it will produce incorrect result.
- Faces on which the boundary condition is not set are treated as having a heat insulation (heat flux is zero).

4-12. Initial Temperature

▶▶ Reference

- For more information, see “Initial Temperature” in Chapter 10 Setting Conditions of Thermal Analysis in the JMAG-Studio User's Manual.
- For more information, see “Initial Temperature” in Chapter 31 “Study Properties and Conditions of Thermal Analysis” in the JMAG-Designer User's Manual.

4-12-1. Function

Defines the temperature at initial time step for transient analysis. It is also possible to set the temperature by using the temperature file (*.temp).

■ Related items

- None

■ Settings

- Targets to set the initial temperature
- Temperature

■ Output items

- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

4-12-2. Important Notes Regarding the Use of This Function

- Always prescribe this condition in transient analysis.
- If this condition is not set, default initial temperature of 0 Celsius is used in the analysis.

4-12-2-1. Initial Temperature Setting Using the Temperature File

The initial temperature can be specified by referring to a temperature file (*.temp).

Conduct the following procedure when using this function.

■ Procedure

This section explains procedures on running an analysis for the thermal analysis input file (*.jcf) "A," and then running an analysis for the thermal analysis input file (*.jcf) "B" which is referring the file "A.temp" that is exported from the input file (*.jcf) "A.".

1. Run an analysis for the input file A.

The temperature file "A.temp" is exported with a JPLOT file when the analysis is completed.

2. Specify settings such as the analysis method for the input file B.

The following should be noted when specifying this setting.

- The same mesh specified in A needs to be used.
- Select [From File] as the specifying method for temperature, and specify A.temp in the initial temperature condition.

3. Run an analysis for the input file B.

4-13. Heat Source (Heat Source Density/Heat Source Amount/Heat Source Density Coefficient)

▶▶ Reference

- For more information, see “Heat Source” in Chapter 10, “Setting Conditions of Thermal Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Heat Source” in Chapter 31, “Setting Conditions of Thermal Analysis” in the JMAG-Designer User's Manual.
- For more information about running a coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

4-13-1. Function

Specifies heat source density or heat generation density.

A heat generation density is specified for each part with a condition for heat source density.

The heat source density is distributed uniformity to faces with a condition set for the heat generation density.

■ Related items

- Time (step)
- Coupled analysis
- Partial model
- Magnetic field analysis: heat source

■ Targets

- Element group
- Element face group
- Solid group
- Solid face group

■ Settings

- Parts and faces to set the heat source
- Setting Heat Source

- Set Heat Source Density to a Constant Value
- Set Heat Source Amount to a Constant Value
- Set Heat Source Density by Point Sequence
- Set Heat Source Amount by Point Sequence
- Coupled analysis (one-way, two-way)

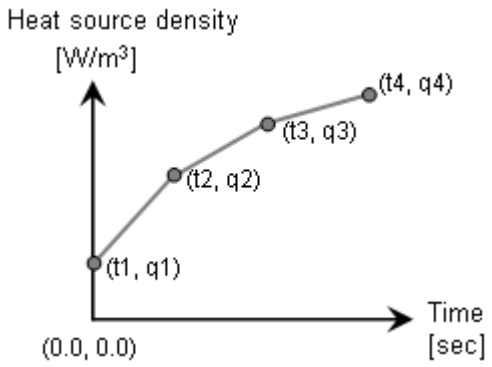
■ Output items

- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)

4-13-2. Important Notes Regarding the Use of This Function

- More than one heat source conditions can be set.
- When specifying the heat source density and heat source amount by point sequence, specify two or more points as shown below.

Time	Heat source density/ Heat source amount/ Heat source density coefficient
t1	q1
t2	q2
t3	q3
t4	q4



Example of heat source density dependent on time

- The Joule loss density (power consumption density) calculated from a magnetic field analysis, electric field analysis, electromagnetic frequency response analysis, and iron loss analysis can be imported as a heat source for a thermal analysis.

  ▶▶ Reference

  For more information, see P.769 "Chapter 10. Coupled Analysis".

- If this condition is set for a face, the shell elements are created on the face and the amount of heat source is specified.
  To view the temperature distribution in the shell elements, the shell elements need to be displayed.
  The face where this condition is set must be very thin, such as a face with strong skin effect due to eddy currents.

4-14. Thermal Circuit

▶▶ Reference

- For more information, see "Equivalent Circuit" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see Chapter 35, "Circuit" in the JMAG-Designer User's Manual.

4-14-1. Function

The circuit can be created for the thermal analysis. Multiple boundary conditions can be set for the terminals in a circuit. These conditions can be set for the thermal analysis using lumped constant.

In JMAG-Studio, the equivalent thermal circuit condition is set.

In JMAG-Designer, the thermal circuit is set.

■ Related items

- Equivalent temperature boundary
- Heat transfer boundary

■ Targets

- Initial temperature
- Thermal resistance
- Heat capacity
- Heat source
- Heat transfer coefficient
- Reference temperature
- Fixed temperature
- The Heat Transfer Boundary condition to be associated with the component
- The Equivalent Temperature Boundary condition to be associated with the component

■ Output items

- Heat flux (W/m^2)
- Terminal Temperature (Celsius)

4-14-2. Important Notes Regarding the Use of This Function

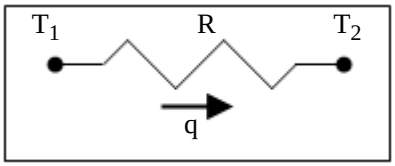
- The following conditions can be allowed to terminals in equivalent circuits.
  - Heat transfer boundary condition
  - Equivalent Temperature Boundary
- In JMAG-Studio, if the [Apply to Equivalent Circuit Condition] radio button selected for the temperature boundary condition or the heat transfer boundary condition, these conditions need to link to all the components by setting the equivalent thermal circuit condition. Unconnected components are displayed in the component list in the [Equivalent Thermal Circuit] dialog box.

4-14-3. Circuit Components

The following circuit components can be used.

■ Thermal resistor component

Defines the resistance to interfere with the heat transfer between two terminals.



$$q = \frac{1}{R}(T_1 - T_2)$$

T_1, T_2:Temperature of terminal

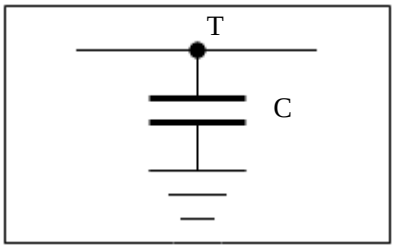
R:Resistance

q:Heat flow

MEMO In a temperature-dependent thermal resistance curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

■ Heat capacitor component

The heat flow into a terminal of the heat capacitor is calculated as the sum of heat flows from terminals connected to it. The rise in temperature after a time step increment is obtained once the temperature of all terminals are calculated.



$$q = C\frac{\Delta T}{\Delta t}$$

T_1, T_2:Temperature of terminal

R:Resistance

C:Heat capacity

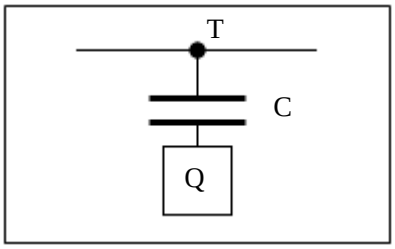
q:Heat flow into the heat capacitor

MEMO In a temperature-dependent heat capacitance curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

■ Heat source component

Applies the amount of heat to the specified terminal.

Set the value of heat source in the terminal of the heat capacitor. The value of heat source cannot be set for other terminals. When a negative value of heat source is set, the terminal becomes endothermic.



$$T = \frac{Q}{C}$$

Q:Heat source amount

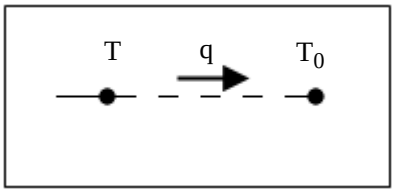
C:Heat capacity

T:Temperature of terminal

MEMO In a temperature-dependent heat source curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

■ Heat transfer component

Defines the heat flowing outside from the specified terminal. Set the heat transfer coefficient and reference temperature.



$$q = h(T - T_0)$$

q:Heat flow

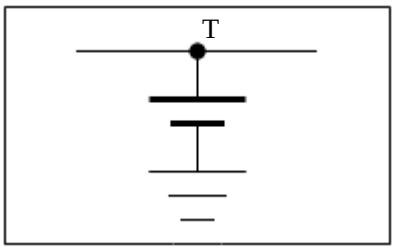
h:Heat transfer coefficient

T_0:Reference temperature

MEMO In a temperature-dependent heat transfer coefficient curve, if the temperature lower than the value specified in the table is referenced during the calculation, each value at the lowest temperature in the table is used. If the temperature higher than the value specified in the table is referenced during the calculation, each value at the highest temperature in the table is used.

■ Fixed temperature component

Maintains the constant temperature for the specified terminal during the analysis.



$$T = T_0$$

T_0:Fixed temperature

4-15. Motion (Rotation/Translation)

▶▶ Reference

- For more information, see "Motion" in Chapter 10 Setting Conditions of Thermal Analysis in the JMAG-Studio User's Manual.
- For more information, see the "Rotation Motion" and "Translation Motion" sections of Chapter 31"Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-15-1. Function

Set the state of motion (velocity) for the element selected. Rotational motion or translational motion can be set.

■ Related items

- None

■ Settings

- Type
  - Rotation motion
  - Translation motion
- Velocity
  - Constant revolution speed
  - Speed point sequence
- Virtual movement flag
- Direction of rotation axis, point on rotation axis *For rotation motion
- Moving direction*For translation motion

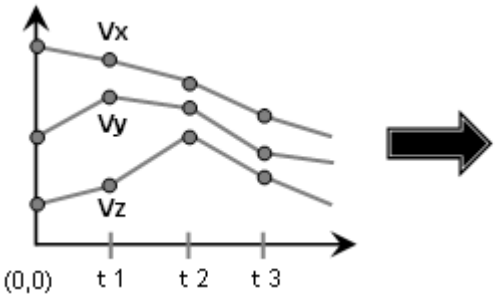
■ Output items

- Displacement (m)

4-15-2. Important Notes Regarding the Use of This Function

- Motion condition can be employed in transient analysis.
- Multiple motion groups can be defined.
- Analysis with no mesh displacement is possible.

■ Velocity table



t	Vx	Vy	Vz
t1	Vx1	Vy1	Vz1
t2	Vx2	Vy2	Vz2
t3	Vx3	Vy3	Vz3
t4	-	-	-
-	-	-	-

- Enter two or more points in the velocity table.
- Intermediate values in the velocity table are generated using linear interpolation between pairs of data points.
- Time specified in table need not necessarily agree with analysis time specified in the step data.

4-16. Average Temperature Calculation

▶▶ Reference

- For more information, see "Average Temperature Calculation" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Average Temperature" in Chapter 31 "Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-16-1. Function

Setting this condition calculates and outputs the average temperature in the specified region. It is used for determining the reference temperature for the FEM coil or resistance component with the temperature-dependent resistance set when coupling with the magnetic field analysis.

■ Related items

- Magnetic field analysis, FEM coil component, Resistance component

■ Settings

- Region to calculate average temperature

■ Output items

- Average temperature (Celsius)

4-16-2. Important Notes Regarding the Use of This Function

- The average temperature specified for this condition and that of materials are output as separate items in the result data.

4-17. Heat Flow Calculation

▶▶ Reference

- For more information, see “Heat Flow Calculation” in Chapter 10 “Setting Conditions of Thermal Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Heat Source” in Chapter 31, “Setting Conditions of Thermal Analysis” in the JMAG-Designer User's Manual.

4-17-1. Function

The heat flow condition is used to calculate heat flow passing through a specified face.

■ Related items

- None

■ Settings

- Calculation region of heat flow
- Inflow Direction

■ Output items

- Amount of heat flow (W)

4-17-2. Important Notes Regarding the Use of This Function

- Multiple groups can be set.

4-18. Contact Thermal Resistance

▶▶ Reference

- For more information, see "Contact Thermal Resistance" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Contact Thermal Resistance" in Chapter 31 "Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-18-1. Function

Considers the thermal resistance between parts.

■ Related items

- None

■ Settings

- Face that thermal resistance is produced
- Thickness/Thermal conductivity (constant or point sequence)
- Thermal Resistance

■ Output items

- Temperature (Celsius)
- Heat flux (W/m^2)
- Heat source density (W/m^3)
- Thermal resistance for each condition (Celsius/W)

4-18-2. Important Notes Regarding the Use of This Function

- Thermal resistance can be set by entering the value, or specifying the thermal conductivity and distance between contact surfaces. When specifying the thermal conductivity, the thermal resistance is determined according to the area of and distance between the contact surfaces and thermal conductivity.
- To specify the thermal conductivity or contact thermal resistance using a point sequence, enter two or more points.
- Set the value of the temperature, which develops in the model during analysis, so that it does not exceed the temperature range defined in the thermal resistance table. If the temperature exceeds the range defined in the table, the value of the end point in the table (the value of the resistance at either the maximum or the minimum temperature) is used.
- If setting the contact thermal resistance condition for a coupled magnetic-thermal analysis, enable the [Use Different Mesh for Each Analysis] option when running the calculation.

4-19. Partial Model

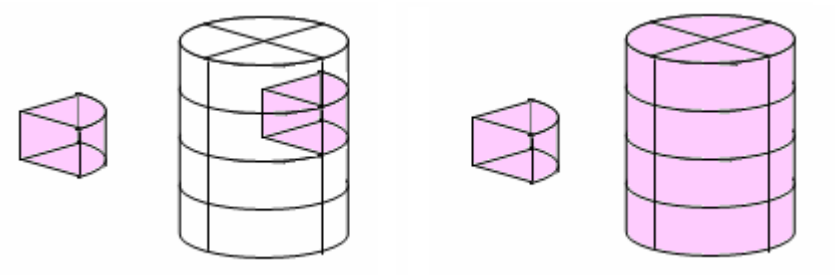
▶▶ Reference

- For more information, see "Partial Model" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Partial Model" in Chapter 31 "Study Properties and Conditions of Thermal Analysis" in the JMAG-Designer User's Manual.

4-19-1. Function

This condition sets the parameters for mapping a partial model on a full model. Setting this condition allows the temperature distribution to be mapped on the magnetic field analysis model if the thermal analysis model is a smaller fraction of the full model than the magnetic field analysis model, for example, 1/4 model for thermal analysis and full model for magnetic field analysis. This condition is used for the coupled thermal-magnetic field analysis.

For example, suppose you want to map the temperature distribution obtained from the thermal analysis with a 1/4 partial model (in the circumferential and axial directions) to the magnetic field analysis full model.



Left: Without the Partial Model condition
Right: With the Partial model condition (rotation/translation)

If the Partial Model condition is not specified, the temperature distribution is mapped only to the part of the magnetic field analysis model that is corresponding to the thermal analysis partial model. By setting the Partial Model condition, it can be mapped from the thermal analysis partial model to the magnetic field analysis full model.

■ Related items

- Coupled thermal-magnetic field analysis

■ Settings

- Periodicity/Symmetry settings on the cross section

  Rotational periodicity:
  - Plane boundary
  - Angle
  - Axis Direction

  Translational periodicity:
  - Plane boundary
  - Distance
  - Direction

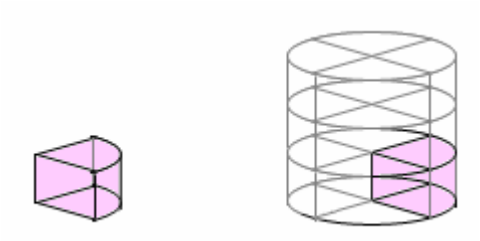
  Mirror:
  - Plane boundary

■ Output items

- N/A

4-19-2. Important Notes Regarding the Use of This Function

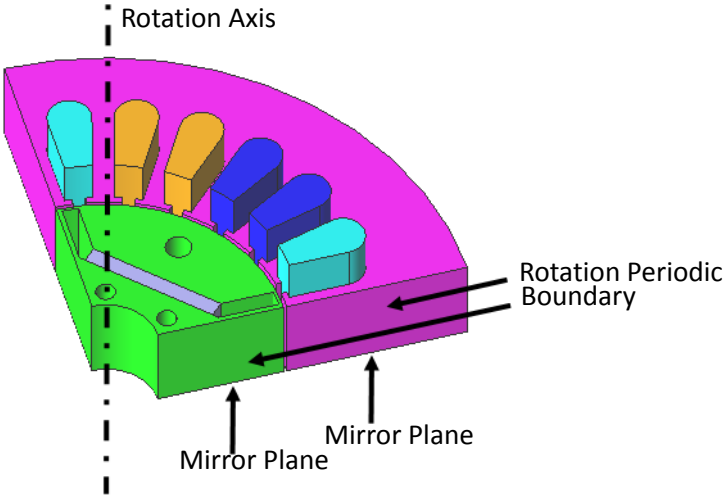
- If you specify both the rotation periodicity and translation periodicity, the partial model is copied in the axis direction.



The following images present the examples of the settings for the partial model condition.

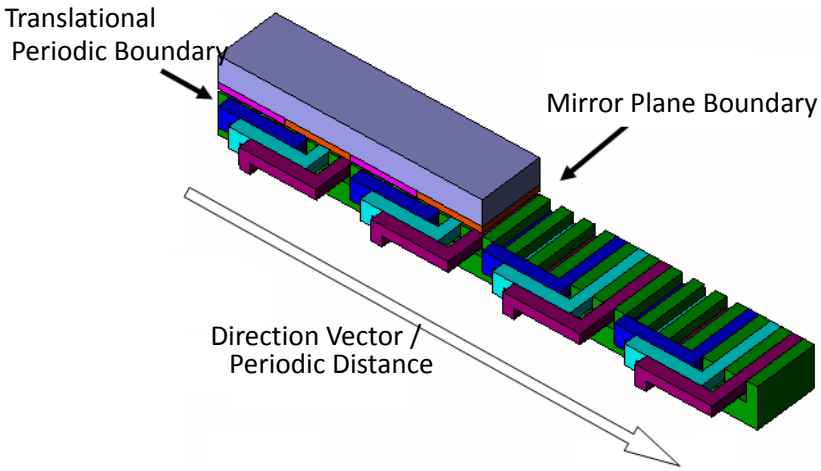
■ Example: Rotation motor

The figure below is a 1/8 partial motor. The Rotational Periodicity (periodic angle of 90 degrees) and Mirror are set on the cross section.



■ Example: Linear motor

The figure below is a 1/2 linear motor partial model. The Translational Periodicity (Periodic Distance is the distance modeled in direction vector on the star) and Mirror are set on the cross section.



4-20. Analysis Type

▶▶ Reference

For more information, see “JMAG-Studio User's Manual,” “Setting Conditions of Thermal Analysis,” “Analysis Control.”

4-20-1. Function

Select the analysis type.

■ Setting

- Analysis type
  - Steady state analysis
  - Transient state analysis

4-20-2. Important Notes Regarding This Function

- Select an analysis type when creating a study for JMAG-Designer.
- Select the analysis type in the thermal analysis control condition for JMAG-Studio. The default value is the steady state analysis.

4-21. Step Control

▶▶ Reference

- For more information, see "Time (Step)" in Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 31 "Study Properties and Conditions of the Thermal Analysis" in the JMAG-Designer User's Manual.

4-21-1. Function

Sets the timing for performing the analysis.

■ Related items

- Motion
- Heat source
- Analysis Control: Analysis Control

■ Settings

- Time (frequency) for each step

  or

- Initial value, end point, and number of divisions

  In this case, analysis time is the defined as:

  Initial value+ (End point-Initial value) / Number of division (Number of steps - 1)

■ Output items

- None

4-21-2. Important Notes Regarding the Use of This Function

- Set the initial value of time to greater than 0.0 sec.
- A time exceeding the time point sequence defined in the heat source condition cannot be set.
- If motion condition is not set, the analysis results are output at the specified time.

- If motion condition is set, the time between 0.0 sec and final time is automatically divided so that displacement for each time interval is shorter than minimum mesh interval.

Reference

For details, see P.448 "4-15. Motion (Rotation/Translation)".

4-22. Two-Way Coupled Analysis

▶▶ Reference

- For more information, see "JMAG-Studio User's Manual," "Setting Conditions of Thermal Analysis," "Analysis Control."
- For more information, see the section in study properties for a thermal analysis in "Chapter 31. Study Properties and Conditions of Thermal Analysis" in the "JMAG-Designer User's Manual."

4-22-1. Function

Specifies a two-way coupled analysis.

▶▶ Reference

For more information about coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

■ Related items

- Heat source condition
- Partial model condition

■ Setting

- Specifies a two-way coupled analysis.

4-23. ICCG

▶▶ Reference

- For more information, see “Analysis Control” of Chapter 10 “Setting Conditions of Thermal Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 31 “Study Properties and Conditions of the Thermal Analysis” in the JMAG-Designer User's Manual.

4-23-1. Function

Defines parameters for controlling the iterative solver.

■ Settings

- Acceleration coefficient
- Convergence tolerance
- Maximum number of iterations

4-23-2. Important Notes Regarding the Use of This Function

■ Acceleration Coefficient

Change this value if after the iteration solutions do not converge. Generally recommended value is about 1.02 - 1.5. Default value is 1.05.

■ Convergence Tolerance

Specify the value at which whether the iteration has converged is determined. Default value is 1.0e-8.

■ Maximum Number of Iterations

If the iteration does not converge, it is terminated when it reaches the maximum number of iteration. The default value is 5000.

4-24. Nonlinear Calculation

▶▶ Reference

- For more information, see "Analysis Control" of Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 31 "Study Properties and Conditions of the Thermal Analysis" in the JMAG-Designer User's Manual.

4-24-1. Function

Defines parameters for nonlinear calculation.

■ Related items

- None

■ Settings

- Maximum number of iterations
- Convergence tolerance

4-24-2. Important Notes Regarding the Use of This Function

4-24-2-1. Maximum Number of Iterations

Specify the maximum number of iterations for nonlinear calculation. Nonlinear calculation is iterated until it is converged at each step.

Nonlinear calculation is terminated when the variation of temperature before and after the iterative calculation reaches the specified convergence tolerance. Nonlinear calculation is also terminated when the specified maximum number of iterations is reached.

- The maximum number of iterations needs to be specified for an analysis with nonlinear materials.
- The default value is 15.

4-24-2-2. Convergence Tolerance

Specify the convergence tolerance for nonlinear calculation.

- A convergence tolerance determines whether the iterative calculation is converged. Nonlinear calculation is converged when the variation of solution reaches the specified convergence tolerance.
- The default value is 1e-3.

4-25. Output Control

▶▶ Reference

- For more information, see "Analysis Control" of Chapter 10 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 31 "Study Properties and Conditions of the Thermal Analysis" in the JMAG-Designer User's Manual.

4-25-1. Function

Defines the items to output as analysis results. The size of a PLOT file and steps to output can also be specified.

■ Related items

- None

■ Settings

- Control output items
  - Temperature (node data)
  - Heat flux (element data)
  - Heat generation density (element data)
- Control the size of plot file *For JMAG-Studio only
- Control the output parts *For JMAG-Designer only
- Control the output interval

4-25-2. Important Notes Regarding the Use of This Function

4-25-2-1. Control Output Items

The following items can be output.

- Temperature
- Heat flux
- Heat generation density

▶▶ Reference

For information about output items, see P.497 "4-30. Output Items of Thermal Analysis".

4-25-2-2. Controlling Output Parts (JMAG-Designer)

In JMAG-Designer, the results of each part can be exported to a result (or jplot) file. The check box of each part needs to be selected to export the results of the part to a jplot file.

Important **If the check box of a material (part) is cleared, all of the element and nodal values belonging to the material (part) are not exported. The check box of a material (part) needs to be selected to view vector and contour distributions in the parts.**

4-25-3. Controlling the File Size (JMAG-Studio)

Specify the [Division of File (MB)] text box.

When results are output exceeding a set value, a PLOT file is automatically divided.

For example, when a calculation is made under the name “Data”, files named “Data.plot001”, “Data. plot002,”... are created. Since the whole of these files compose a set of results of an analysis, do not delete any part of them.

4-25-3-1. Control Output Interval

The size of a result file (*.jplot, *.plot) can be minimized by specifying steps to output. There are three ways to specify steps.

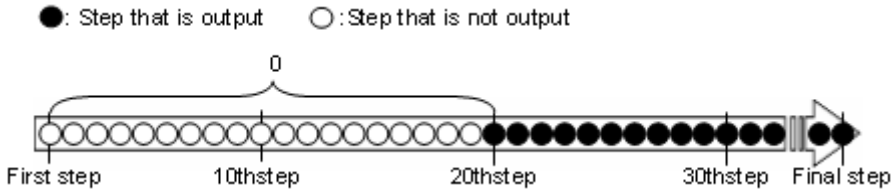
■ All steps

Select this to outputs the results for all of the steps.

■ From step

Select this to outputs the results for all steps from the specified step.

For example, if you enter “20”, the results for step 20 and all steps thereafter will be output as shown below.



■ Specify interval

Specify the step intervals using a point sequence.

- The output interval can be defined more specifically. For example, you can output the results every 5 steps up to step 20, and from step 21 every 2 steps.
- If you set the output interval to "0", the steps before the specified step are not output.
- If you set the output interval to "1", all steps before the specified step are output.
- The step specified with a table and the steps thereafter are all output.
- The step IDs should be specified in ascending order in a table.

▶▶ Reference
For examples of the output interval table, see P.320 "3-52-2-4. Controlling Output Intervals".

4-26. Parallel Computing: Shared Memory Multiprocessing (SMP)

▶▶ Reference

For more information, see “Study Properties” in Chapter 31 “Study Properties and Conditions of the Thermal Analysis” in the JMAG-Designer User's Manual.

4-26-1. Function

The shared memory multiprocessor (SMP) is used to perform iterative calculations on multiple CPUs (cores) using parallel computing in a thermal analysis. An increase in calculation speed based on parallelism can be expected when large scale and non-linear materials are used.

4-26-2. Important Notes Regarding the Use of This Function

The following should be noted when using the SMP.

- The degree of parallelism can be selected from 2, 4, and 8.

4-26-2-1. License Type

The license type requires one of the solver licenses (HT) and the SMP licenses as many as the degree of parallelism.

For example, the number of licenses required for parallel computing in a thermal analysis varies as follows:

- Two parallel processing: 1 HT solver license and 1 SMP license
- Four parallel processing: 1 HT solver license and 2 SMP licenses
- Eight parallel processing: 1 HT solver license and 4 SMP licenses.

MEMO The solver licenses for each analysis are as follows:
HT ---- Thermal analysis (3D)

MEMO Please contact your local distributor for more information about recommended hardware environment for running parallel computing.

MEMO The environment variables need to be set when using parallel computing (distributed memory multiprocessing (DMP)). Please contact your distributor for support.

4-27. Controlling User Subroutine

▶▶ Reference

- For more information, see "Analysis Control" of Chapter 10 Setting Conditions of Thermal Analysis in the JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 31 "Study Properties and Conditions of the Thermal Analysis" in the JMAG-Designer User's Manual.

4-27-1. Function

Selects the user subroutine to use in the analysis.

User subroutine is a program that a user creates to express a function that JMAG cannot realize by itself and to perform a calculation. The user subroutine must be written in Fortran or C++.

■ Related items

- User subroutine

■ Settings

- User subroutine DLL
- User subroutine

4-27-2. Important Notes Regarding the Use of This Function

- Specify the user subroutine DLL to be used in the analysis in the [Select User Subroutine DLL] group box of the [User Subroutine] dialog box.

  When using a user subroutine DLL on a remote machine for the analysis run on the remote system, the user subroutine DLL needs to be added to the list.
- The user subroutine DLL to be used in the analysis must be stored in the following locations:
  - Windows: JMAG Installation Directory\usrsub\heat
  - Linux /works/solver/mod/heat/
- If you want to distribute the calculation that uses the user subroutine DLL, place the user subroutine DLL if using a Windows machine, or shared library if using a Linux machine, on all CPU nodes in the CPU group to be used for the distributed calculation.

- The user subroutine is enabled by selecting the subroutine check box in the [User Subroutine] group box of the [User Subroutine] dialog box.

  The user subroutine that does not have any check box on this pane becomes available under the other conditions, such as the occasion when the user-defined material was chosen in the materials property.

4-28. Restart

▶▶ Reference

- For more information, see "Analysis Control" in Chapter 11 "Setting Conditions of Thermal Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Running the analysis using one study" in Chapter 40 Running Analysis in the JMAG-Designer User's Manual.

4-28-1. Function

Calculation is re-executed using results from a previous analysis as the initial values.

If the size of the PLOT file becomes too large in a single analysis, the output can be divided by step.

In transient analysis, if it is found that the steady state has not yet been reached after the analysis, the analysis can be continued run by using this function.

■ Related items

- Time step/step control
- Analysis control

■ Settings

- Restart

4-29. User Subroutine

Subroutines are available as user subroutines in JMAG. These subroutines can be customized.

▶▶ Reference

For more information about required environment, usage, and exporting report files, see P.330 "3-56. User Subroutine".

4-29-1. Types of User Subroutine

■ ***dtuser***

When analysis that puts motion into consideration is conducted, designate the time division of the analysis.

■ ***usrstp***

Allows the time to be passed to magnetic field analysis in the process of coupled analysis with magnetic field analysis.

■ ***usrhtc***

Designate the coefficient of heat transfer.

■ ***usrhrd***

Designates the coefficient of radiation heat transfer.

■ ***ucond***

It is possible to designate the nonlinear anisotropic thermal conductivity. By setting a multiple temperature thermal conductivity table against a single physical property, it is possible to handle heterogeneous materials (materials of which the material properties are in a different state against an identical property.).

■ **uspeci**

Corresponding to heterogeneous materials, it is possible to designate a multiple temperature specific heat table against a single physical property.

■ *flgph*

Specifies whether or not to use user subroutine *phase1* and *phase2* to determine the physical properties formed by heterogeneous materials.

■ *phase1*

Determines the status for each element at the initial status (first step) when conducting coupled analysis with electromagnetic waves on an analysis object containing heterogeneous material. On and after the second step, the flag indicating the status for each element written out at the previous step (referring to the material ID of material forming heterogeneous material with the same physical property) is read in.

■ *phase2*

When performing coupled analysis with electromagnetic waves on an analysis object containing heterogeneous material, the status flag is written out after calculation by subroutines defined and created by the user.

■ *udnsty*

Defines the density.

■ *quser*

Specifies heat source based on the function which can be expressed by either element or position of the coordinate system.

4-29-2. dtuser

Reference
For more information, see P.448 "4-15. Motion (Rotation/Translation)".

4-29-2-1. Settings

- Defines time increment.

4-29-2-2. Arguments

Variable	Details	I/O
vx(numnp)	Nodal velocity in X direction	I
vy(numnp)	Nodal velocity in Y direction	I
vz(numnp)	Nodal velocity in Z direction	I
numnp	Number of nodes in the model	I
almin	Minimum mesh length in the model	I
dt	Time increment	O

4-29-2-3. Example

In this example, derived time increment is multiplied by 5 in the program.

■ **Program (Fortran)**

```
      subroutine dtuser (vx,vy,vz,numnp,almin,dt)
      implicit real*8 (a-h,o-z)
c
      dimension vx(numnp),vy(numnp),vz(numnp)
c
c
      dt = dt * 5.0
c
c
      return
      end
```

4-29-3. usrstp

Reference

For more information about coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

4-29-3-1. Settings

- Passes time to magnetic field analysis in coupled analysis with magnetic field.
- For JMAG-Studio, to use this subroutine, select the [usrstp] check box in the [User Subroutine DLL] group box for the analysis control condition of thermal analysis.

4-29-3-2. Arguments

Variable	Details	I/O
time	Time	I

4-29-3-3. Example

In this example, the time is passed to the file of htime of device number 100.

■ **Program (Fortran)**

```
          subroutine usrstp(time)
         implicit real*8 (a-h,o-z)
c
          open(100, file='htime', status='unknown',
form='formatted')
         rewind(100)
c
         write(100,'(1pd20.10)') time
c
         close(100)
c
         return
         end
```

4-29-4. usrhtc

▶▶ Reference
For information, see P.431 "4-9. Heat Transfer Boundary".

4-29-4-1. Settings

- Defines a heat transfer coefficient.
- To use this subroutine, select the [Use User Subroutine] radio button when setting the heat transfer boundary condition.

4-29-4-2. Arguments

Variable	Details	I/O
nowelm	Element ID of heat transfer boundary	I
nowfac	Element surface ID of heat transfer boundary	I
idim	Size of XYZ array (=3)	I
xyz(idim)	Coordinates of the integral point on the element surface of heat transfer boundary	I
time	Analysis time	I
temp	Temperature on the element surface of heat transfer boundary	I
htc	Heat transfer coefficient	O

4-29-4-3. Example

In this example, the heat transfer coefficient is set to 1.

■ Program (Fortran)

```
subroutine  usrhtc
          (nowelm, nowfac, idim, xyz, time, temp, htc)
c
      integer   nowelm, nowfac, idim
      real*8                              xyz(idim)
      real*8                                      time, temp, htc
c
       htc = 1.0d0
c
      end
```

4-29-5. usrhrd

Reference
For more information, see P.434 "4-10. Radiation Heat Transfer Boundary".

4-29-5-1. Settings

- Defines a radiant heat transfer coefficient.
- To use this subroutine, select the [Use User Subroutine] radio button when setting the radiation heat transfer boundary condition.

4-29-5-2. Arguments

Variable	Details	I/O
nowelm	Element ID of radiation heat transfer boundary	I
nowfac	Element surface ID of radiation heat transfer boundary	I
idim	Size of XYZ array (=3)	I
xyz(idim)	Coordinates of the integral point on the element surface of radiation heat transfer boundary	I
time	Analysis time	I
temp	Temperature on the element surface of radiation heat transfer boundary	I
htc	Radiation heat transfer coefficient	O

4-29-5-3. Example

In this example, the emissivity (gray coefficient) is set to 1, and the radiation heat transfer coefficient is determined by multiplying the emissivity by the Stefan-Boltzmann constant.

■ **Program (Fortran)**

```
      subroutine  usrhrd
            (nowelm, nowfac, idim, xyz, time, temp, htc)
c
      integer   nowelm, nowfac, idim
      real*8                            xyz(idim)
      real*8                                   time, temp, htc
c     sigma : Stefan-Boltzmann constant
      real*8      sigma
      parameter (sigma = 5.670508685704303d-8)
c     ems   : emissivity coefficient
      real*8      ems
c
      ems = 1.0d0
c
c     htc must be multiplied Stefan-Boltzmann constant

       htc = sigma * ems
c
      end
```

4-29-6. ucond

Reference
For more information, see P.423 "4-4. Material Property for Each Part".

4-29-6-1. Settings

- Defines an aerotropic thermal conductivity with temperature dependency. Specifying multiple temperature-thermal conductivity tables enables support of heterogeneous materials.
  "Heterogeneous materials" are materials for which different physical property tables need to be referenced in response to the status in the analysis, as the material represents different properties with part on it.
  "Status" represents the phase of matter such as solid or liquid.
- For homogeneous isotropic materials with temperature dependency, select the value in the [Material List] dialog box.
- This function is contained in the file usrmat.f (usrmat.cpp).
- For JMAG-Studio, to use this subroutine, select the [User Subroutine] radio button in the [Electric Properties] tab in the [Material] dialog box. For JMAG-Designer, to use this subroutine, select [User Subroutine ucond] in the [Temperature Properties] tab in the [Material Editor] dialog box.

4-29-6-2. Arguments

Variable	Details	I/O
ielm	Element ID	I
mtyp	Material ID of element	I
iflgph	Flag of phase (*1)	I
temp	Temperature of the integral point of the element	I
ukapp(3)	Thermal conductivity	O

**1 iflgph* is not used unless iph = 0,1 is set with *phase*.f.

4-29-6-3. Example

In this example, thermal conductivity is made 1.

■ **Program (Fortran)**

```
      subroutine ucond(ielm,mtyp,iflgph,temp,ukapp)
c
      implicit real*8 (a-h,o-z)
c
      dimension ukapp(3)
c
      ukapp(1) = 1.0d0
      ukapp(2) = 1.0d0
      ukapp(3) = 1.0d0
c
      return
      end
```

4-29-7. uspeci

Reference
For more information, see P.423 "4-4. Material Property for Each Part".

4-29-7-1. Settings

- Defines a temperature-dependent temperature-specific heat table that contains heterogeneous material.
  "Heterogeneous materials" are materials for which different physical property tables need to be referenced in response to the status in the analysis as the material represents different properties with part on it.
  "Status" represents the phase of matter such as solid or liquid.
- For homogeneous materials with temperature dependency, specify the value in the [Material List] dialog box.
- This function is contained in the file usrmat.f (usrmat.cpp).
- For JMAG-Studio, to use this subroutine, select the [User Subroutine] radio button in the [Specific Heat] group box in the [Material] dialog box. For JMAG-Designer, to use this subroutine, select [User Subroutine uspeci] in the [Temperature Properties] tab in the [Material Editor] dialog box.

4-29-7-2. Arguments

Variable	Details	I/O
ielm	Element ID	I
mtyp	Material ID of element	I
dnety0	Density set by input.	I
iflgph	Flag of Phase (*1)	I
temp	Temperature of the integral point of element	I
rouc	Specific heat	O

*1 *iflgph* is not used unless iph = 0,1 is set with *phase*.f.

4-29-7-3. Example

In this example, the specific heat is a value set by input.

■ **Program (Fortran)**

```
      subroutine uspeci(ielm,mtyp,dnsty0,iflgph,temp,rouc)
c
c     ielm ielm-th element
c     iflgph Material status flag
c     temp Temperature at ielm-th element
c     rouc Specific Heat
c
      implicit real*8 (a-h,o-z)
c
      include '../comn/cntrl.com'
      include '../comn/file.com'
c
      parameter (maxtyp = 2)
      parameter (maxp   = 4)
c
      dimension  tabltp(maxp,maxtyp),tablsp(maxp,maxtyp)
c
     data tabltp /-1.0000000000E+02 , 2.0000000000E+02 ,
    #              2.0010000000E+02 , 1.0000000000E+06 ,
c
    #             -1.0000000000E+02 , 2.0000000000E+02 ,
    #              2.0010000000E+02 , 1.0000000000E+06 /
c
      data tablsp / 3.0000000000E+02 , 3.0000000000E+02 ,
    #              3.0000000000E+02 , 3.0000000000E+02 ,
c
    #              3.0000000000E+02 , 3.0000000000E+02 ,
    #              3.0000000000E+02 , 3.0000000000E+02  /
c*******************************************************
      if     (iflgph .eq. 0) then
        jj = 1
elseif (iflgph .eq. 1) then
        jj = 2
      endif
c
      do 1000 ii=2,maxp
        if (temp.lt.tabltp(ii,jj)) then
          dspdtp =
     #             (tablsp(ii,jj) - tablsp(ii-1,jj))
     #            /(tabltp(ii,jj) - tabltp(ii-1,jj))
          rouc   = (temp - tabltp(ii-1,jj))*dspdtp
     #           + tablsp(ii-1,jj)
        end if
 1000 continue
c*******************************************************c
      return
```

```
end
```

4-29-8. flgph

Reference
For more information, see P.423 "4-4. Material Property for Each Part".

4-29-8-1. Settings

- This user subroutine enables routines, *phase1* and *phase2*, that specify the status flag (here the status represents the phase of matter such as solid or liquid) for each element in the analysis in which heterogeneous material is contained. It defines a temperature-dependent temperature-specific heat table that contains heterogeneous material.
  "Heterogeneous materials" are materials for which different physical property tables need to be referenced in response to the status in the analysis as the material represents different properties with part on it.
  "Status" represents the phase of matter such as solid or liquid. If the flag specified in this routine is not turned on, you cannot access the routine to specify the status flag.
- Even if the flag is turned on, you are unable to access the routine to specify the status flag if the temperature-specific heat property of phase change material does not reference the user subroutine *uspeci*.
- For JMAG-Studio, to use this subroutine, select the [flgph] check box in the [User Subroutine DLL] dialog box for the analysis control condition of thermal analysis.

4-29-8-2. Arguments

Variable	Details	I/O
iph	0: Do not use the routine to specify the status flag 1: Use the routine to specify status the flag	I /O

4-29-8-3. Example

■ **Program (Fortran)**

```
      subroutine flgph(iph)
c
      implicit real*8 (a-h,o-z)
c
c  This routine determines if the user uses the following
subroutine.
c  Please specify the flag "iph".
c
c  iph = 0 : User subroutine is not available.
c  iph = 1 : User subroutine is available.
c
      iph=1
c
      return
      end
```

4-29-9. phase1

4-29-9-1. Settings

- This subroutine is used in coupled analysis with heat and electromagnetic waves for objects containing heterogeneous material.
  "Heterogeneous materials" are materials for which different physical property tables need to be referenced in response to the status in the analysis as the material represents different properties with part on it.
  "Status" represents the phase of matter such as solid or liquid. Status of the material can be defined by specifying flag values according to the status of each element.
- This routine writes out the status flag at the initial step of the coupled analysis, and on and after the second step, the hstate file in which status flags are described for each element by the user in phase2. There are no limits on the values of the status flags, but it is recommended that they be set to 0, 1, 2, and so on, according to the phase.
- Note that when using this routine, thermal conductivity and specific heat should also be specified by the user subroutine.
- For JMAG-Studio, to use this subroutine, select the [phase1] check box in the [User Subroutine DLL] dialog box for the analysis control condition of thermal analysis.

4-29-9-2. Arguments

Variable	Details	I/O
istep	Current number of analysis steps	I
time	Current analysis time	I
numnp	Number of nodes of model	I
numel	Number of elements of model	I
maxnod	Maximum number of nodes within an element	I
lm	Global node ID determined by element ID and local node ID of each element	I
mtyp	Material ID	I
numj	Number of elements to which heat source is given	I
numqfl	Number of elements to which joule loss density is given	I
qnow	Heat source density for each element at analysis time	I
qold	Same as above	I
nodph	Array of nodes for user	I
elmph	Array of elements for user	I
iflgph	Status flag for each element	I

4-29-9-3. Example

In this example, the initial value (0) of the status flag is defined at the zero step in the thermal analysis, and the flag value at the previous step is then read in from the status flag file “hstate”.

■ **Program (Fortran)**

```
      subroutine phase1(istep,time,numnp,numel,maxnod,lm,
     #                  mtyp,numj,numqfl,qnow,qold,
     #                  nodph,elmph,
     #                  iflgph)
c
c     istep     Step value for the current step
c     time      Time of the current step
c     numnp     Number of total nodes
c     numel     Number of total elements
c     lm        Global node ID determined by element ID and
local node
c     mtyp      Material number for each element
c     numqfl    Number of elements for lossive material
c               Currently it is the same as that of numel.
c     qnow      Jouleloss density of each element
c     qold      Jouleloss density of each element
c     nodph     For user
c     elmph     For user
c     iflgph    Status flag for each element
c
      implicit real*8 (a-h,o-z)
c
      include '../comn/cntrl.com'
      include '../comn/file.com'
c
      dimension lm(maxnod,numnp),mtyp(numel),
     #          qnow(numj),qold(numj),
     #          nodph(numnp),elmph(numel),iflgph(numel)
c
c     Definition of initial status value for iplgph
c     Please do not change the following file format.
c
    if (istep.eq.0) then
open(ifl24,file='hstate',status='unknown',form='formatted')
          do 10 ielm = 1,numel
            iflgph(ielm) = 0
   10     continue
          rewind ifl24
          write(ifl24,'(i10)') numel
          write(ifl24,'(8i5)') (iflgph(ielm),ielm=1,numel)
        close(ifl24)
          go to 200
endif
c
```

```
      idumy = 0
      fdumy = 0.0
c
c     Reading the status file.
c     Please do not change the following file format.
c

open(ifl24,file='hstate',status='unknown',form='formatted')
        rewind ifl24
        read(ifl24,'(i10)') idummy
        read(ifl24,'(8i5)')(iflgph(ielm),ielm=1,numel)
      close(ifl24)
c
  200 continue
c
      return
      end
```

4-29-10. phase2

4-29-10-1. Settings

- In *phase2*, the final thermodynamic status is defined at a given time using a user-defined equation (created in *phase2*) based on the status flags for each element, which were loaded in phase 1. The status flags are also updated for the temperatures obtained for each element.
- The updated hstate file is passed to the electromagnetic wave analysis, and in the electromagnetic wave analysis, physical property values are defined in response to the status of each element before analysis is run.
- Refer to the following sample for details of the of hstate file format.
- For JMAG-Studio, to use this subroutine, select the [phase2] check box in the [User Subroutine DLL] dialog box for the analysis control condition of thermal analysis.

4-29-10-2. Arguments

Variable	Details	I/O
numnp	Number of nodes of model	I
numel	Number of elements of model	I
maxnod	Maximum number of nodes within an element	I
idim	Not used	I
coord	Coordinates of node	I
lm	Global node ID determined by element ID and local node ID of each element	I
numqfl	Number of elements to which joule loss density is given	I
qfl	Joule loss density	I
istep	Current analysis step	I
time	Current analysis time	I
numat	Same as above	I
ckap0	Not used	I
dnst0	Not used	I
rocp0	Not used	I
mtyp	Material ID	I
iflgph	Status flag for each element	I/O
fai	Nodal temperature at current step	I/O
faiold	Nodal temperature at previous step	O
faiusr	User-defined nodal temperature	O

4-29-10-3. Example

In this example, two temperature-specific heat tables are provided. When the temperature of each element exceeds the critical temperature tc, the status flag is changed from 0 to 1, and the status is then written out to the status flag file "hstate".

■ **Program (Fortran)**

```
        subroutine phase2(numnp,numel,maxnod,idim,coord,lm,
     #           numqfl,qfl,istep,time,
c===============In(Material)=================

     #           numat,ckap0,dnst0,rocp0,mtyp,
c===============In/Out=====================

     #           iflgph,fai,faiold,faiusr)
c
c     numnp     Number of total nodes
c     numel Number of total elements
c     idim      Dimenssion(=3)
c     coord     Global coordinate for each node
c     lm        Global node ID determined by element ID and
local node
c     numqfl    Number of elements for loss material
c     qfl       Joule loss density of each element
c     istep     Step value for the current step
c     time      Current Time
c     numat     Number of material type
c     ckap0     Not used
c     dnst0     Not used
c     rocp0     Not used
c     mtyp      Material number for each element
c     iflgph    Material status for phase transition material
c     faiusr    Node temperature defined by user
c     faiold    Node temperature for previous step
c
      implicit real*8 (a-h,o-z)
c
      include '../comn/cntrl.com'
      include '../comn/file.com'
c
      dimension coord(idim,numnp),lm(maxnod,numel),qfl(numqfl)
      dimension
ckap0(numat),dnst0(numat),rocp0(numat),mtyp(numel)
      dimension
iflgph(numel),fai(numnp),faiold(numnp),faiusr(numnp)
c
c---------------------For user-----------------------
c     Dummy model for check
c
c     Initialization
c
```

```
        do 490 inod = 1,numnp
        faiusr(inod) = 0.0
  490   continue
c
c     Getting an average temperature of nodes on the same
c     element and defining the material state.
c
c       tc is the critical temperature.
c
        tc = 50.0
c
        write(ifl01,'(1pd20.10)') time
        do 500 ielm = 1,numel
          avtemp = 0.0
          do 510 j = 1,8
            inode = lm(j,ielm)
          avtemp = avtemp + 0.125*fai(inode)
            write(ifl01,'(1p8d20.10)') fai(inode)
  510     continue
          write(ifl01,'(i10,1pd20.10)')ielm,avtemp
          if (avtemp.gt.tc) then
            iflgph(ielm) = 1
          else
            iflgph(ielm) = 0
          endif
  500   continue
c
c-------------------------------------------------------
c     Please do not change the following file format.
c     (ifl23 = hloss2)


open(ifl23,file='hloss2',status='unknown',form='formatted')
        fdumy = 0.0
        idumy = 0
        rewind ifl23
        write(ifl23,'(1pd20.10,i10)') fdumy,numel
        do 900 ielm = 1,numel
          write(ifl23,'(i10,1pd20.10)') ielm,fdumy
  900   continue
      close(ifl23)
c
c     Please do not change the following file format.
c     Status flag file for HT and WV (ifl24 = hstate)


open(ifl24,file='hstate',status='unknown',form='formatted')
        rewind ifl24
        write(ifl24,'(i10)') numel
        write(ifl24,'(8i5)') (iflgph(ielm),ielm=1,numel)
      close(ifl24)
c
       do 1000 inod = 1,numnp
```

```
                fai(inod) = fai(inod) + faiusr(inod)
 1000   continue
c
      return
      end
```

4-29-11. udnsty

4-29-11-1. Settings

- Defines density.
- This function is contained in the file usrmat.f (usrmat.cpp).
- For JMAG-Studio, to use this subroutine, select the [User Subroutine] radio button in the [Density] group box in the [Material] dialog box. For JMAG-Designer, to use this subroutine, select [User Subroutine udnsty] in the [Physical Properties] tab in the [Material Editor] dialog box.

4-29-11-2. Arguments

Variable	Details	I/O
ielm	Element ID	I
mtyp	Material ID of element	I
dnety0	Density set by input	I
iflgph	Flag of phase (Note 1)	I
temp	Temperature of the integral point of element	I
dnsty	Density	O

- *1 *iflgph* is not used unless iph = 0,1 is set with phase.f.

4-29-11-3. Example

In this example, the density is a value set by input.

■ **Program (Fortran)**

```
      subroutine udnsty(ielm,mtyp,dnsty0,iflgph,temp,dnsty)
c
      implicit real*8 (a-h, o-z)
c
      dnsty = dnsty0
c
      return
      end
```

4-29-12. quser

4-29-12-1. Settings

- Specifies heat source based on the function which can be expressed by either element or position of the coordinate system. Time-dependent functions can be used.
- For JMAG-Studio, to use this subroutine, select the [quser] check box in the [User Subroutine DLL] dialog box for the analysis control condition of thermal analysis.

4-29-12-2. Arguments

Variable	Details	I/O
iqusr	User subroutine flag	I/O
ne	Element number	I
imat	Element property number	I
time	Time	I
pc	Barycentric coordinates of element. pc(1), pc(2), pc(3) represent X, Y, Z coordinates respectively.	I
qext	Element ne heat source density (W/m^3)	O

4-29-12-3. Example

A time-dependent heat source in physical media is shown below. It shows the heat source spreading evenly or varying at a wavelength of 250μm by the setting of time table for each coefficient.

$$qA1 = Cobj \times rIobj + Cref \times rIref + 2\sqrt{Cobj \times rIobj \times Cref \times rIref \cos(Ck \times rkx \times pc(1))}$$

$$qA2 = Cgap \times Cass \times rIass$$

$$qext = qA1 + qA2$$

The coefficients that represent heat source qA1 and qA2 have time dependency as follows.

- Heat Source qA1

  Timet_1o0 - t_1oE: Cobj=1.0, Everything else = 0

  Timet_1r0 - t_1rE: Cref=1.0, Everything else = 0
- Heat Source qA2

  Timet_20 - t_2E: Cgap=1.0, Cass=1.0 Everything else = 0

  Note that the time is defined as t_20<t_1r0<t_1o0<t_1rE<t_1oE<t_2E. rIobj, rIref, rIass all represent heat source density (W/m^3).

■ Program (Fortran)

```
c234567890

============================================================
c This subroutine defines an arbitrary heat source
c function defined by user.
c
c  iqusr(OUT)  Flag for this subroutine 0(Not use)/1(Use)
c     ne(IN)   Element number
c   imat(IN)   Material number for the element ne
c   time(IN)   Current time
c     pc(IN)   Center of coordinate for the element ne
c  qext(OUT)  Heat source density
c
c============================================================
c===== TYPE A =====
      subroutine quser(iqusr,ne,imat,time,pc,qext)
c
      implicit real*8 (a-h,o-z)
c
      dimension pc(3)
c
      iqusr=1
c
      pie=4.0d0*atan(1.0d0)
c     Wave Length in the Material (m)
      rlamd=2.5d-7
c     Wave Vector (rad/m)
      rkx=2.0d0*pie/rlamd
c     Coefficience
      Cobj =0.d0
      Cref =0.d0
      Cgap =0.d0
      Cass =0.d0
      Ck   =1.d0
c     Intensity (W/m3)
      rIobj =8.d14
      rIref =8.d14
      rIass =8.d14
c     Time Table for Switchiing (sec)
      t_1o0=2.d-7
      t_1oE=4.d-7
      t_1r0=1.d-7
      t_1rE=3.d-7
      t_20 =0.d0
      t_2E =1.d-5
c
      if (imat.eq.1) then
        if (time.ge.t_1o0 .and. time.le.t_1oE) then
          Cobj=1.d0
        endif
        if (time.ge.t_1r0 .and. time.le.t_1rE) then
```

```
          Cref=1.d0
        endif
        if (time.ge.t_20 .and. time.le.t_2E) then
          Cgap=1.d0
        Cass=1.d0
        endif
        qA1 = Cobj*rIobj+Cref*rIref
     #      +
2.d0*sqrt(Cobj*rIobj*Cref*rIref)*cos(Ck*rkx*pc(1))
        qA2 = Cgap*Cass*rIass
        qext = qA1+qA2
      else
        qext=0.0d0
      endif
  100 continue
c
      return
      end
```

4-30. Output Items of Thermal Analysis

Thermal analysis module (HT) can display the following as analysis results:

MEMO Each of the export items are displayed in units that are applied to an analysis model in JMAG-Designer.

Output item	Unit	Type	Object	Note
Temperature	Celsius	scalar	node	
Heat flux	W/m^2	vector	element (solid)	
Heat flow	W	step	element face	Output when the heat flow calculation condition is set.
Displacement	m	vector	node	The amount of displacement from the previous step at each node is obtained when the motion condition is set
Heat source density	W/m^3	scalar	element	
Amount of heat source	W	step	-	
Average temperature	Celsius	step	-	
Terminal temperature	Celsius	step	-	Temperature of terminal in the equivalent circuit are displayed.
Average temperature (condition)	Celsius	step	-	Output when the average temperature calculation condition is set.
Total displacement	Rotation motion: deg Translational motion: m	step	-	Output when [Virtual Movement] is not set for motion condition.

Chapter 5. Function of Electric Field Analysis

This chapter describes in detail how to set the material properties and analysis conditions used in electric field analyses.

▶▶ Reference

For the operation procedures, see the JMAG-Studio User's Manual or the JMAG-Designer User's Manual.

5-1. Overview of Functions

The electric field analysis module employs the finite element method (FEM) with unknown electric potential. Electric field analysis is performed with the following modules:

5-1-1. EL (Electric Field Analysis Module)

The electric field analysis (static analysis, frequency response analysis) and the current distribution analyses can be performed.

■ Electric field analysis (static analysis)

- When permittivity, electric potential, or electric charge is specified, electric potential and electric field intensity are calculated.

■ Electric field analysis (frequency response analysis)

- When permittivity, electric conductivity, electric potential, and frequency are specified, electric potential, electric field intensity, and current distribution are calculated.

■ Current distribution analysis

- When electric conductivity and electric potential are specified, electric potential and current distribution are calculated.
- This analysis can be coupled with a thermal analysis (HT).
- User subroutines can be employed in the analysis.

▶▶ Reference

- For the details of the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

- For the details of the user subroutine, see P.540 "5-29. User Subroutine (Current Distribution Analysis: EL)".

5-1-2. CH (Charge Distribution Analysis Module)

- The electric field analysis (EL: electric field analysis module) coupled with the transfer and diffusion of particles is performed.
- User subroutines must be employed in an analysis of the transfer and diffusion of charged particles.

Important **The charge distribution analysis is only available in JMAG-Studio.**

MEMO This concerns hexahedron elements only. Moreover transformed hexahedron elements cannot be used.

▶▶ Reference

For the details of the user subroutine, see P.546 "5-30. User Subroutine (Charge Distribution Analysis: CH) (JMAG-Studio)".

5-2. Conditions of Electric Field Analysis (JMAG-Designer)

The analysis conditions supported by JMAG-Designer are listed below. Some conditions may not be available depending on the module used.

Analysis conditions	Module*1		
	EL	EL-FQ	EL-CUR
Material			
Material Properties	⊙	⊙	⊙
Boundary			
Rotation periodic boundary	O	O	O
Translation periodic boundary	O	O	O
Symmetry boundary	O	O	O
Natural boundary	O	O	O
Electric potential boundary	O	O	O
Electric field boundary	O	O	X
Current boundary	X	X	O
Current density boundary	X	X	O
Output			
Surface charge density	O	O	O
Force	O	O	O
Current	X	X	O
Modeling			
Conductor	O	O	X
Insulation	X	X	O
Charge			
Face charge	O	X	X
Volume charge	O	X	X
Link			
Temperature distribution	X	X	O

Analysis conditions	Module*1		
	EL	EL-FQ	EL-CUR
Study properties			
Frequency control	X	O	X
ICCG	O	O	O
Nonlinear calculation	O	O	O
Output control	O	O	O
Coupled analysis	X	O	O

⊙: Required O: Optional X: Not available

*1 Type of analysis:

- EL: Electric field analysis (static analysis)
- EL-FQ: Electric field analysis (frequency response analysis)
- EL-CUR: Current distribution analysis

5-3. Conditions of Electric Field Analysis (JMAG-Studio)

The analysis conditions supported by JMAG-Studio are listed below.

Analysis conditions	Module*1			
	EL	EL-FQ	EL-CUR	CH
Material Properties	⊙	⊙	⊙	⊙
Electric potential boundary (EL)	⊙	⊙	⊙	X
Electric potential boundary (CH)	X	X	X	⊙
Electric field boundary/Current density boundary	O	O	O	O
Periodic boundary	O	O	O	O
Conductor	O	O	O	O
Force calculation using electric field	O	O	O	O
Surface charge distribution calculation	O	O	O	O
Face charge density	O	O	X	O
Volume charge density	O	O	X	O
Control data of charge distribution analysis (particle information)	X	X	X	O
Initial particle density (CH)	X	X	X	O
Current calculation	X	X	O	X
Time and frequency	⊙	⊙	⊙	⊙
Analysis control	⊙	⊙	⊙	⊙
Control data of charge distribution analysis	⊙	⊙	⊙	⊙
User Subroutine				
usrdns (initial number of particles, initial charge)*2	X	X	X	⊙
velusr (velocity of each particle)*2	X	X	X	⊙
regusr (user items)*2	X	X	X	⊙
dtuser (time increment)*2	X	X	X	⊙
setdns (change of number of particles, spatial charge)*2	X	X	X	⊙
bmuser*2	X	X	X	⊙
setdns*2	X	X	X	⊙

⊙: Required O: Optional X: Not available

*1 Type of analysis:

- EL: Electric field analysis (static analysis)
- EL-FQ: Electric field analysis (frequency response analysis)
- EL-CUR: Current distribution analysis
- CH: Charge distribution analysis

*2 Set items using user subroutines.

5-4. Material Property for Each Part

5-4-1. Specifying the Particle Flight Region (Charge Distribution Analysis)

- When the material property region is a particle flight region, select the [ON] radio button of the [Particle Flight Region] group box.

5-4-2. Post Output (JMAG-Studio)

- Only materials for which the [Output] radio button of the [Post Output] group box is selected are displayed in the result window.
- Only items related to materials for which [Output] radio button is selected are exported to a plot file.
- The [Output] radio button is selected in default.

MEMO The output can be set by part for JMAG-Designer.

5-5. Periodic Boundary (Rotation/ Translation)

▶▶ Reference

- For more information, see "Periodic Boundary" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG Studio User's Manual.
- For more information, see "Translation Periodic Boundary" or "Rotation Periodic Boundary" in Chapter 33 "Study Properties and Conditions of Electric Field Analysis," in the JMAG-Designer User's Manual.

5-5-1. Function

When an analysis target has periodicity about an axis, the periodic boundary condition is used to reduce the size of a model to analyze.

■ Related items

- None

■ Settings

- Nodes on the first periodic boundary surface

  MEMO The first periodic boundary plane refers to the periodic boundary plane that is in a counterclockwise direction of the two periodic boundary planes in a rotation periodic model.

- Specifying the rotational/translation periodic boundary face.

  Rotational periodic boundary
  - A point on rotation axis
  - Direction of rotation axis
  - Angle of rotation
  - Specifying periodic/antiperiodic boundary

  Translation periodic boundary
  - Direction of translation periodic
  - A point on rotation axis
  - Periodic distance
  - Specifying periodic/antiperiodic boundary

■ Available modules

- All of the modules of electric field analysis (EL, CH)* CH is available in JMAG-Studio only.

■ Output items

- None

5-5-2. Important Notes Regarding the Use of This Function

- A rotation periodic boundary for axial reverse is not supported.

5-5-2-1. Target Boundary

- Select nodes on the first periodic boundary surface of an analysis model.
- The first periodic boundary surface is a boundary surface having the smaller angle in the circumferential direction at both ends of a model with symmetry relative to the axis of rotation. The second periodic boundary surface is a boundary surface having the larger angle.
- JMAG-Studio automatically determines the nodes on the second periodic boundary surface (or edge). Nodes on the second periodic boundary surface (or edge) must correspond to the nodes on the first periodic boundary surface (or edge).

5-5-2-2. Periodic Boundary/Anti-periodic Boundary

- In the case of the periodic boundary condition, when a model is rotated by the periodic angle, the size and direction of certain physical quantities (e.g. temperature distribution) are unchanged before and after the rotation (see Fig. 1).

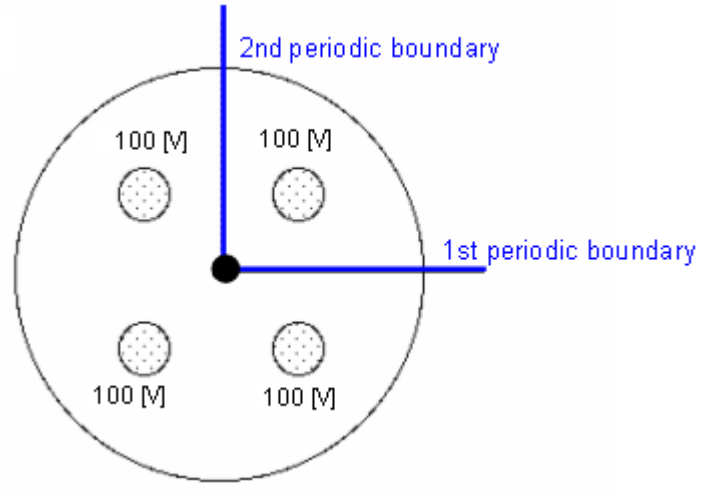


Fig.1 Periodic boundary

- In the case of the antiperiodic boundary condition, when a model is rotated by periodic angle, the size of certain physical quantities (e.g. magnetic flux density) is unchanged but its direction is reversed before and after the rotation (see Fig. 2).

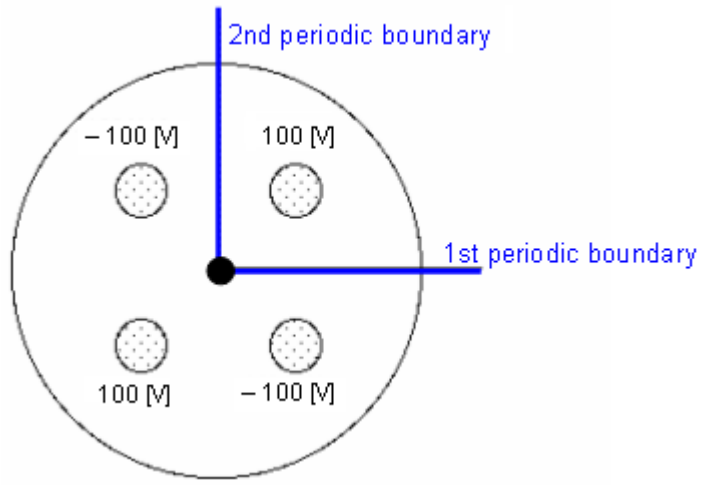


Fig.2 Antiperiodic boundary

5-5-2-3. Rotational Periodic Boundary

- "Point on rotation axis" means the coordinate for central section of the model, e.g., the coordinate is (0,0,0) for a model which is periodic around Z-axis centering the origin.
- "Direction of rotation axis" means a vector passing through center of rotation, e.g., if an axial vector is set to Z-axis, the axial vector is (0,0,1).
- "Periodic angle" means the angle of the center of the analysis model. For a 1/4 model the periodic angle is 90deg.

5-5-2-4. Translational Periodic Boundary

- "Translation direction" is the vector representing the direction of translation. For example, if the boundary has a translational periodicity in the X-axis direction, the direction is (1, 0, 0).
- For the translational periodic boundary, enter the periodic distance instead of the periodic angle. For example, if it has a periodicity every 30 cm, the periodic distance is 0.3 m.

5-6. Symmetry Boundary/Natural Boundary (JMAG-Designer)

▶▶ Reference

For more information, see "Symmetry Boundary" in Chapter 33, "Study Properties and Conditions of Electric Field Analysis," in the JMAG-Designer User's Manual.

5-6-1. Function

The symmetry boundary condition and the natural boundary condition are set to the analysis model.

The electric flux flows parallel to the face the symmetry boundary condition is set. The electric flux flows perpendicular to the face the natural boundary is set.

■ Related items

- N/A

■ Settings

- Natural boundary face

■ Available modules

- EL (static analysis)
- EL (frequency response analysis)
- EL (current distribution analysis)

■ Output items

- N/A

5-6-2. Important Notes Regarding the Use of This Function

- The symmetry boundary condition is set to a face the electric field is completely shielded.
- Faces the symmetry boundary condition is not set are treated as a natural boundary face the electric flux flows perpendicularly.

5-7. Electric Potential Boundary (EL)

▶▶ Reference

- For more information, see “Electric Potential Boundary (EL)” in Chapter 11, “Setting Conditions of Electric Field Analysis” in JMAG Studio User's Manual.
- For more information, see “Electric Field Analysis” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-7-1. Function

Sets the electric potential for parts, faces, edges, and vertex that is specified.

■ Related items

- Electric potential boundary (CH)

■ Settings

- Target
  - Node in JMAG-Studio only
  - Parts, faces, edges, and vertices in JMAG-Designer only
- Electric potential

■ Available modules

- EL (static analysis)
- EL (frequency response analysis)
- EL (current distribution analysis)

■ Output items

- Electric potential (V) (static analysis, current distribution analysis)
- Electric potential (real part, imaginary part) (V) (frequency response analysis)
- Electric field (V/m) (static analysis, frequency response analysis)
- Current density (A/m^2) (current distribution analysis, frequency response analysis)

5-7-2. Important Notes Regarding the Use of This Function

- This condition is used when the electric potential for the parts, faces, edges, and vertex that is specified is constant.
- Always define this condition when to performing a static analysis, frequency response analysis, or current distribution analysis.
- This condition cannot be set in a charge distribution analysis.

5-8. Electric Potential Boundary (CH)

▶▶ Reference

For more information, see “Electric Potential Boundary (CH)” in Chapter 11, “Setting Conditions of Electric Field Analysis” in JMAG-Studio User's Manual.

5-8-1. Function

Sets the electric potential for the node that is specified.

■ Related items

- Electric potential boundary (EL)

■ Settings

- Target
  - Node in JMAG-Studio only
- Electric potential table

■ Available modules

- CH (charge distribution analysis)* JMAG-Studio only

■ Output items

- Electric potential (V)
- Electric field (V/m)

5-8-2. Important Notes Regarding the Use of This Function

- This condition is used when electric potential specified for the node is constant.
- This condition must be set in a charge distribution analysis.
- This condition cannot be specified in an electric field analysis (static analysis, frequency response analysis) or a current distribution analysis.
- This condition can be set more than once.
- This condition specifies time-dependent electric potential.
- Define an electric potential table using point sequence. An electric potential table must contain two or more pairs of data points.

5-9. Electric Field Boundary

▶▶ Reference

- For more information, see "Electric Field/Current Density Boundary" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG-Studio User's Manual.
- For more information, see "Electric Field Boundary" in Chapter 33, "Study Properties and Conditions of Electric Field Analysis," in the JMAG-Designer User's Manual.

5-9-1. Function

Sets the electric field of the inflow face or the outflow face in an electric charge.

■ Related items

- None

■ Settings

- Target (face)
- Normal component of electric field (electric field boundary)

■ Available modules

- EL (static analysis)
- EL (frequency response analysis)
- CH (charge distribution analysis)* JMAG-Studio only

■ Output items

- Voltage (V)
- Electric field(V/m)
- Current density (A/m^2)

5-9-2. Important Notes Regarding the Use of This Function

- Specify this setting when the electric field is distributed in uniformity for the face that is specified.
- Specify the normal component of the electric field.

5-10. Current Boundary (JMAG-Designer)

▶▶ Reference

For more information, see “Current Analysis” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-10-1. Function

Sets the amount of current in the inflow face or outflow face.

■ Related items

- N/A

■ Settings

- Boundary face
- Normal component of current

■ Available modules

- EL (current distribution analysis)

■ Output items

- Electric Potential
- Electric Field
- Current Density

5-10-2. Important Notes Regarding the Use of This Function

- This function is used when the distribution of current is in uniformity on the face that is specified.
- Specify the normal component of the current.
- The current density boundary condition is used when specifying the amount of current in the inflow face or outflow face using the electric current density.

5-11. Current Density Boundary

▶▶ Reference

- For more information, see "Chapter 11. Setting Conditions of Electric Field Analysis," "Electric Field/Current Density Boundary" in the JMAG-Studio User's Manual.
- For more information, see "Chapter 33. Study Properties and Conditions of Electric Field Analysis," "Current Density Boundary" in the JMAG-Designer User's Manual.

5-11-1. Function

The amount of current in the inflow face or outflow face can be set in electric current density.

■ Related items

- Current boundary

■ Setting

- Target (face)
- Normal component of current density

■ Modules that can be used with this function

- EL (current distribution analysis)

■ Output items

- Electric potential
- Electric field
- Current density

5-11-2. Important Notes Regarding the Use of This Function

- This function is used when uniformity of distributed current on the face is specified.
- Specify the normal component of the current density.
- The current boundary condition is used when directly specifying the amount of current in the inflow face or outflow face.

5-12. Force Calculation by Electric Field

▶▶ Reference

- For more information, see “Force Calculation” in Chapter 11, “Setting Conditions of Electric Field Analysis” in JMAG-Studio User's Manual.
- For more information, see “Force” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis,” in the JMAG-Designer User's Manual.

5-12-1. Function

The force calculation condition is set to calculate the force induced by the electric field acting on the specified elements and nodes.

■ Related items

- None

■ Targets

- Element groups

■ Settings

- None

■ Available modules

- All of the modules of electric field analysis (EL, CH)* CH is available in JMAG-Studio only

■ Output items

- Nodal force (N)
- Force by electric field (N)

5-12-2. Important Notes Regarding the Use of This Function

- For the specified elements of a model, resultant force and each nodal force are calculated.

5-13. Surface Charge Distribution Calculation (Surface Charge Density)

▶▶ Reference

- For more information, see "Surface Charge Distribution Calculation" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG Studio User's Manual.
- For more information, see "Surface Charge" in Chapter 33, "Study Properties and Conditions of Electric Field Analysis", in the JMAG-Designer User's Manual.

5-13-1. Function

The surface charge distribution calculation condition defines the distribution of a charge density and a total density of the specified surface.

■ Related items

- None

■ Targets

- Element faces

■ Settings

- None

■ Available modules

- All of the modules of electric field analysis (EL, CH)* CH is available in JMAG-Studio only

■ Output items

- Charge density (C/m^2)
- Charge(C)

5-13-2. Important Notes Regarding the Use of This Function

- This condition is used to calculate unknown charge density. The charge density cannot be specified with this condition.
- Most outer surface of the analysis model cannot be selected.

5-14. Current Calculation

▶▶ Reference

- For more information, see “Current Calculation” in Chapter 11, “Setting Conditions of Electric Field Analysis” in JMAG- Studio User's Manual.
- For more information, see “Current” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-14-1. Function

The current calculation condition defines a total amount of current flowing through the cross section of a specified area of an analysis model.

■ Related items

- None

■ Settings

- Calculation region for current

■ Available modules

- EL (current distribution analysis)

■ Output items

- Current (A)

5-14-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.

- When selecting an element face, specify one of two element faces as a cross section.
  If two faces are selected, this condition cannot be calculated correctly.

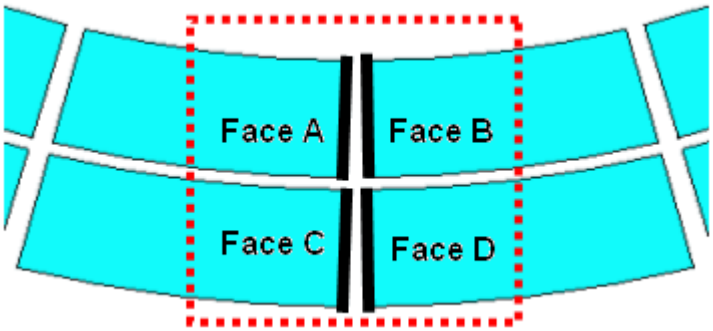


Fig.1 Enlarged elements

As shown in Fig. 1, if the area surrounded by red dotted lines is selected by dragging a mouse pointer, both faces (element face A and element face B, element face C and element face D) which elements are adjacent are selected. As a result, accurate calculation results cannot be obtained.

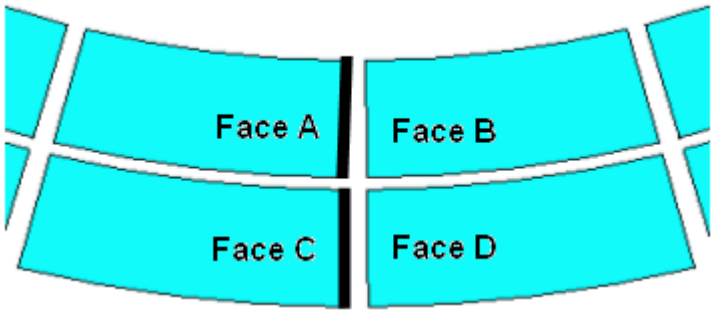


Fig.2 Enlarged elements

As shown in Fig. 2, select only faces of one side (element face A and element face C) of the area in which the elements are adjacent.

5-15. Conductor

▶▶ Reference

- For more information, see “Conductor” in Chapter 11, “Setting Conditions of Electric Field Analysis” in JMAG-Studio User's Manual.
- For more information, see “Conductor” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-15-1. Function

The conductor condition applies the same electric potential to each area.

■ Related items

- Electric potential boundary (EL)
- Electric potential boundary (CH)

■ Targets

- Elements

■ Settings

- None

■ Available modules

- All of the modules of electric field analysis (EL, CH) * CH is available in JMAG-Studio only

■ Output items

- Voltage (V)

5-15-2. Important Notes Regarding the Use of This Function

- When the electric potential is specified for the nodes of elements with the electric potential boundary condition, this condition cannot be correctly taken into account during an analysis.

5-16. Insulation

▶▶ Reference

For more information, see "Insulation" in Chapter 33, "Study Properties and Conditions of Electric Field Analysis" in the JMAG-Designer User's Manual.

5-16-1. Function

The insulation condition is used to include the effect of electric insulation in the analysis.

■ Related items

- N/A

■ Settings

- Insulated face

■ Available modules

- EL (current distribution analysis)

5-16-2. Important Notes Regarding the Use of This Function

- This condition specifies a face to separate a conductor to take into account electric insulation. However, the mesh does not change before and after the analysis.

5-17. Face Charge (Face Charge Density)

▶▶ Reference

- For more information, see “Surface Charge” in Chapter 11, “Setting Conditions of Electric Field Analysis” in JMAG Studio User's Manual.
- For more information, see “Face Charge” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-17-1. Function

The surface charge condition defines a surface charge density for the specified element faces.

■ Related items

- Volume charge condition

■ Targets

- Element faces

■ Settings

- Surface charge density

■ Available modules

- EL (static analysis)
- EL (frequency response analysis)
- CH (charge distribution analysis)* JMAG-Studio only

■ Output items

- Voltage (V)
- Electric field (V/m)

5-17-2. Important Notes Regarding the Use of This Function

- This condition is used to calculate electric potential by specifying a surface charge density for element faces. An unknown charge density cannot be calculated with this condition.

5-18. Volume Charge (Charge Density)

▶▶ Reference

- For more information, see "Volume Charge" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG-Studio User's Manual.
- For more information, see "Volume Charge" in Chapter 33, "Study Properties and Conditions of Electric Field Analysis" in the JMAG-Designer User's Manual.

5-18-1. Function

The volume charge condition defines a charge density for the specified elements.

■ Related items

- Face charge condition

■ Settings

- Part to define charge density
- Charge density

■ Available modules

- EL (static analysis)
- EL (frequency response analysis)
- CH (charge distribution analysis)* JMAG-Studio only

■ Output items

- Voltage (V)
- Electric field (V/m)

5-18-2. Important Notes Regarding the Use of This Function

- This condition is used to specify a charge density. An unknown charge density cannot be calculated.

5-19. Temperature Distribution (JMAG-Designer)

▶▶ Reference

- For more information, see “Temperature Distribution” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.
- For more information, see P.769 "Chapter 10. Coupled Analysis".

5-19-1. Function

An analysis is run using materials with temperature dependency by referring to thermal analysis results.

■ Related items

- Material properties
- Two-way coupled analysis

■ Setting

- Coupling type
  - One-way
  - Two-way
- Result file of thermal analysis for referencing (one-way)

■ Modules that can be used with this function

- EL (current distribution analysis)

5-19-2. Important Notes Regarding the Use of This Function

- This function is specified when using material data with temperature dependent electric property.

5-20. Control Data of Charge Distribution Analysis (Particle Data) (JMAG-Studio)

▶▶ Reference

For more information, see "Control Data of Charge Distribution Analysis" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG-Studio User's Manual.

5-20-1. Function

The control data of charge distribution analysis condition defines the number of particle types, transition coefficient matrix between particles, and diffusion coefficients.

■ Related items

- Initial particle density condition (CH)
- User subroutine (CH)

■ Targets

- Analysis model

■ Settings

- Transition coefficient matrix between particles
- Diffusion coefficient of each particle

■ Available modules

- CH (charge distribution analysis)* JMAG-Studio only

■ Output items

- Particle density (pcs/m^3)

5-20-2. Important Notes Regarding the Use of This Function

- Always set this condition for a charge distribution analysis.
- In the transition matrix, use a point sequence to specify a transition rate at which a particle transits from one type (state) to a different type (state) under the condition that there are multiple particle types (states).

  When there is only one type (state) of particle and no transition occurs, specify 1 as the number of particle types.

5-21. Initial Particle Density (JMAG-Studio)

▶▶ Reference
For more information, see "Initial Particle Density" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG-Studio User's Manual.

5-21-1. Function

The initial particle density condition defines an initial particle density for each particle.

■ Related items

- User subroutine: *usrdns*

■ Settings

- Initial particle density
- Particle type

■ Available modules

- CH (charge distribution analysis)

■ Output items

- Particle density (pcs/m^3)

5-21-2. Important Notes Regarding the Use of This Function

- Initial particle densities can also be set with the user subroutine *usrdns*.
- When there are multiple types of particles, specify a serial number for each particle from 1 in ascending order.

  For example, when there are three types, specify a number, 1, 2, or 3 for each type.

5-22. Analysis Type

▶▶ Reference

For more information, see “JMAG-Studio User's Manual,” “Chapter 11. Setting Conditions of Electric Field Analysis,” “Analysis Control.”

5-22-1. Function

Select the analysis type.

■ Setting

- Analysis type
  - Electric field analysis (static analysis)
  - Electric field analysis (frequency response analysis)
  - Current distribution analysis
  - Charge distribution analysis

■ Modules that can be used with this function

- All of the modules of electric field analysis (EL, CH) * CH is available in JMAG-Studio only.

5-22-2. Important Notes Regarding the Use of This Function

- Select an analysis type when creating a study for JMAG-Designer.
  Select the analysis type in the electric field analysis control condition for JMAG-Studio. The default value is the electric field analysis (static analysis).
  The materials that can be set in the material dialog differ depending on the analysis type that is selected.

5-23. Step Control / Frequency Control

▶▶ Reference

- For more information, see "Time" in Chapter 11, "Setting Conditions of Electric Field Analysis" in JMAG-Studio User's Manual.
- For more information, see "Study Properties" in Chapter 33, "Study Properties and Conditions of Electric Field Analysis" in the JMAG-Designer User's Manual.

5-23-1. Function

The step condition sets the timing (or frequency) for performing the analysis.

In a charge distribution analysis, intervals of electric field analyses and output intervals of analysis results are set.

■ Related items

- None

■ Settings

- Time (static analysis, current distribution analysis)
- Frequency (frequency response analysis)
- Time, intervals of electric field analyses, and output intervals of analysis results (charge distribution analysis)

■ Available modules

- All of the modules of electric field analysis (EL, CH)* CH is available in JMAG-Studio only

■ Output items

- None

5-23-2. Important Notes Regarding the Use of This Function

- For JMAG-Studio, set the initial time to 0.0 in an electric field analysis or current distribution analysis.
- Set frequency in a frequency response electric field analysis. More than one frequency can be set.
- For a charge distribution analysis, set time (Te), intervals of electric field analyses, and output intervals of analysis results.
- In a charge distribution analysis, the term “time” means the analysis completion time. The analysis is run for an integral number of times, in which the integral number is the integer closest to the quotient obtained by dividing a specified time by the time step (automatically calculated in the program or specified in the user subroutine).

  If the intervals of electric field analyses is set to “10”, the electric field analysis is performed once per ten steps for the specified time (Te).

  If the interval of analysis results is set to “100”, the analysis result is exported once per 100 steps for the specified time (Te).

  More than one point sequence can be created by setting different values for time/frequency, frequency of electric field analysis, and output frequency of analysis results.

5-23-2-1. Number of Analysis Steps

- When multiple frequencies are used to perform electric field analysis (frequency response analysis), set the number of analysis steps, which is equivalent to the number of frequencies. The default is 1.
- Use the default value when running an electric field analysis (static analysis), a current distribution analysis, and a charge distribution analysis in JMAG-Studio.

MEMO A text box to set the number of steps is not available in the electric field analysis (static analysis) and the current distribution analysis in JMAG-Designer.

5-24. Two-Way Coupled Analysis

▶▶ Reference

For more information, see "JMAG-Designer User's Manual," "Chapter 33. Study Properties and Conditions of Electric Field Analysis," "Study Properties."

5-24-1. Function

Specifies a two-way coupled analysis.

■ Related items

- Material (temperature dependent material)
- Temperature distribution condition

■ Setting

- Two-way coupled analysis flag

▶▶ Reference

For information about the coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

5-25. ICCG

▶▶ Reference

- For more information, see “Analysis Control: Setting of ICCG Method” in Chapter 11, “Setting Conditions of Electric Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-25-1. Function

Specifies the parameters for the iterative solver of the simultaneous linear equations.

■ Related items

- N/A

■ Settings

- Acceleration coefficient
- Convergence tolerance
- Divergence tolerance
- Maximum iterations

■ Available modules

- All of the modules of electric field analysis (EL, CH) * CH is available in JMAG-Studio only

■ Output items

- N/A

5-25-2. Important Notes Regarding the Use of This Functions

5-25-2-1. Acceleration Coefficient

Change this value if the iteration solutions do not converge. Generally, the recommended value is between 1.02 and 1.5. Default value is 1.05.

5-25-2-2. Convergence Tolerance

Specify the value that determines whether the iteration is converged. Default value is 1.0e-8.

5-25-2-3. Maximum Number of Iterations

If the iteration does not converge, it is terminated when it reaches the maximum number of iterations. Change this value if the iteration dose not converge with the default value. The default value is 5000.

5-26. Nonlinear Calculation

▶▶ Reference

- For more information, see “Analysis Control: Setting of ICCG Method” in Chapter 11, “Setting Conditions of Electric Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 33, “Study Properties and Conditions of Electric Field Analysis” in the JMAG-Designer User's Manual.

5-26-1. Function

Species the parameters for the nonlinear calculation.

■ Related items

- N/A

■ Settings

- Maximum nonlinear iterations
- Convergence tolerance

■ Available modules

- All of the modules of electric field analysis (EL, CH) * CH is available in JMAG-Studio only.

5-26-2. Important Notes Regarding the Use of This Function

5-26-2-1. Number of Nonlinear Iterations

- The nonlinear iterative calculation continues until the number of iterations reaches the specified maximum number of iterations or the convergence value reaches the specified convergence tolerance.
- Specify the maximum number of iterations only for a nonlinear analysis.
- The default value is 15.
- For the nonlinear iterative calculation, the successive iteration method is used.

5-26-2-2. Convergence Tolerance

Define a convergence tolerance for a nonlinear calculation.

- The nonlinear iterative calculation continues until the number of iterations reaches the specified maximum number of iterations or the convergence value reaches the specified convergence tolerance.
- Define the number of iterations only for a nonlinear analysis.
- The default value is 1e- 3.

5-27. Post Output

▶▶ Reference

- For more information, see, “Setting Conditions of Electric Field Analysis” in the JMAG-Studio User's Manual.
- For more information, see “Study Properties” in Chapter 33 “Study Properties and Conditions of the Electric Field Analysis” in the JMAG-Designer User's Manual.

5-27-1. Function

This setting defines the items to export to a result file.

■ Related items

- None

■ Settings

- Electric potential (node data)
- Electric field (element data)
- Particle density (node data)
- Nodal charge (node data)
- Current (element data)
- Surface charge (element data)
- Nodal force (node data)
- Internal charge (element data)
- Power consumption (element data)
- Thermal electromotive force loss (element data)
- Total loss (element data)

▶▶ Reference

For more information, see P.569 "5-31. Output Items of Electric Field Analysis".

■ Available modules

- All of the modules of electric field analysis (EL, CH)* CH is available in JMAG-Studio only

5-27-2. Important Notes Regarding the Use of This Function

- In a static analysis and frequency response analysis, if the [Internal Charge (element data)] check box is selected, the following items are exported. If there is a nodal charge, [Nodal Charge] is output.
  - If there is a line charge, [Line Charge Density] is output.
  - If there is a surface charge, [Surface Charge Density] is output.
  - If there is a polarization vector, [Polarization Vector] is output.

  These output items are used for an analysis when a plot file of an electric field analysis is specified for the integral method calculator (CALCB) tool.

5-28. Parallel Computing: Shared Memory Multiprocessing (SMP)

▶▶ Reference

For more information, see “Study Properties” in Chapter 33 “Study Properties and Conditions of the Electric Field Analysis” in the JMAG-Designer User's Manual.

5-28-1. Function

The shared memory multiprocessor (SMP) is used to perform iterative calculations on multiple CPUs (cores) using parallel computing in a electric field analysis. An increase in calculation speed based on parallelism can be expected when large scale and non-linear materials are used.

5-28-2. Important Notes Regarding the Use of This Function

The following should be noted when using the SMP.

- The degree of parallelism can be selected from 2, 4, and 8.

5-28-2-1. License Type

The license type requires one of the solver licenses (EL) and the SMP licenses as many as the degree of parallelism.

For example, the number of licenses required for parallel computing in a electric field analysis varies as follows:

- Two parallel processing: 1 EL solver license and 1 SMP license
- Four parallel processing: 1 EL solver license and 2 SMP licenses
- Eight parallel processing: 1 EL solver license and 4 SMP licenses.

MEMO The solver licenses for each analysis are as follows:
EL ---- Electric field analysis (3D)

MEMO Please contact your local distributor for more information about recommended hardware environment for running parallel computing.

MEMO The environment variables need to be set when using parallel computing (distributed memory multiprocessing (DMP)). Please contact your distributor for support.

5-29. User Subroutine (Current Distribution Analysis: EL)

5-29-1. Types of User Subroutines

■ ***cndusr***

Defines the electric conductivity matrix.

■ ***usesee***

Defines the usage of Seebeck coefficient.

■ ***usrsee***

Defines the Seebeck coefficient of elements.

5-29-2. Required Environment for User Subroutine

- Fortran is supported as a programming language. Development environments where user subroutines can be built are the following.
  - Fortran
    - Microsoft Visual Studio 2010 and Intel Fortran Composer XE 2011 (Intel Visual Fortran Compiler XE 12.1)

5-29-3. Notes on Using User Subroutines

- The following device numbers are reserved. Use unused device numbers for user subroutines.

  Reserved device numbers: 1, 2, 6, 10, 11, 20, 21, 22, 23, 24, 55
- The electric potential, electric field, and current of each step are exported as analysis results.
- DLL files used in JMAG-Studio cannot be used in JMAG-Designer. The files need to be rebuilt in a new compiler.

5-29-4. Reconfiguration of JMAG

To use user subroutines, the total module needs to be reconfigured after Fortran program file (*.f) is rewritten arbitrarily.

5-29-4-1. Intel Visual Fortran 12.1

1. **Make sure that VisualStudio2010 is installed to the machine.**

   In case it is not installed, a module cannot be created.

2. **Open the Fortran program file (*.f) in the following directory, rewrite the contents, and save.**

   `(JMAG installation directory)\solver\mod\elec\usrsub`

   Do not change the directory and file name of the Fortran program file.
   It is recommend to create a backup file for the Fortran program file prior to rewriting.

3. **Start VisualStudio2010.**

4. **Open usrsub.sln.**

   usrsub.sln is located in the same directory in step 2.
   Select [File] > [Open] > [Project] from the menu bar, and then open usrsub.sln.

5. **When right-clicking [Project] of [Solution Explorer], if [Extract Compaq visual Fortran Project Items] is displayed in the shortcut menu, select it, and then specify usrsub.dsp.**

   usrsub.dsp is also located in the same directory in step 2. [Extract Compaq Visual Fortran Project Items] is displayed in the menu for the first time to compile the program with your machine.
   If [Extract Compaq Visual Fortran Project Items] is not displayed in the menu, proceed to step 6.

6. **Execute a reconfiguration of the module (elec.exe).**

   Select [Build] > [Build of solution] or [Build] > [Rebuild of solution] from the menu bar, and execute a reconfiguration.
   When the reconfiguration is executed, a new module (elec.exe) is created in the following directory.
   `(JMAG installation directory)\solver\mod\elec\mod`

7. **Place the module (elec.exe) created in step 6 in the installation directory of JMAG-Studio.**

   At this time, it is recommend to place a different name on the original elec.exe and save it.

8. **Execute an analysis with JMAG-Studio.**

Important **When creating a user subroutine for 64 bit using VisualStudio2010, perform the following steps to set up the development environment.**
1. Open the project with VisualStudio2010.
2. Click the [Solution Platform] list (labeled as Win32), and if available, select [x64] to enable the build configuration.
If "64 bit" does not appear in the list, add it to the list as follows.
3. Select [Configuration Manager] from the [Solution Platform] list.
4. Select [New] from the [Active Solution Platform] list.
5. Select [x64] from the [Enter or Select New Platform] list.
6. [x64] can be selected from the [Solution Platform] list in step 2.

Important **The build environment for Intel 64 needs to be added by selecting [Custom Install] when installing Visual Studio.**

Important **When installing Visual Studio and Intel Fortran on the same machine, make sure that they are installed in the correct order (first Visual Studio, and then Intel Fortran), and they are set up for building the 64-bit module. For more information, see the manual of each toolkit.**

5-29-4-2. JMAG Linux Module

1. **Save the Fortran program file (*f) that was edited using the procedures in "Intel Visual Fortran 12.1 "(described above) in the following directory.**

   ```
   (JMAG installation directory)solver\mod\elec\usrsub
   ```

2. **Replace it under the directory of step 1.**

3. **Execute [%make] to start a reconfiguration.**

   ```
   Here "%" means the prompt.
   ```

5-29-5. cndusr

5-29-5-1. Settings

- *cndusr* defines the electric conductivity matrix.
- When using *cndusr*, enter the ID (negative value) of the argument flag in the [Material] dialog box.

5-29-5-2. Arguments

Variable	Details	I/O
ne	Element ID	I
mtyp	Material ID of an element	I
iflag	Flag of the material ID (=-10)	I
sigma(3,3)	Electric conductivity matrix	O
ev(3)	Electric field at the integral point of an element	I
temp	Temperature at the integral point of an element	I

5-29-5-3. Example

In this example, the electric conductivity matrix is made the diagonal matrix of 1e6.

■ Program

```
      subroutine cndusr(ne,mtyp,iflag,sigma,ev,temp)
c
      implicit real*8 (a-h,o-z)
      dimension sigma(3,3),ev(3)
c
      val = 1.d6
      sigma(1,1) = val
      sigma(1,2) = 0.0d0
      sigma(1,3) = 0.0d0
      sigma(2,1) = 0.0d0
      sigma(2,2) = val
      sigma(2,3) = 0.0d0
      sigma(3,1) = 0.0d0
      sigma(3,2) = 0.0d0
      sigma(3,3) = valc
      return
      end
```

5-29-6. usesee

5-29-6-1. Settings

- *usesee* defines the use of Seebeck coefficient

5-29-6-2. Arguments

Variable	Details	I/O
lseeb	Flag of the Seebeck coefficient	O

5-29-6-3. Example

In this example, the flag of the Seebeck coefficient is set to 1, and the Seebeck coefficient can be set to elements. If the flag is set to 0, the Seebeck coefficient cannot be used.

```
      subroutine  usesee (lseeb)
c
      implicitr real*8 (a-h,o-z)
c
c     set Seebeck calculation flag
c     (if lseeb /=0) Seebeck calculation executed,
c                           but usrsee must be set.
!     lseeb = 0
      lseeb = 1
c
      return
      end
```

5-29-7. usrsee

5-29-7-1. Settings

- *usrsee* defines the Seebeck coefficient of elements.

5-29-7-2. Arguments

Variable	Details	I/O
ne	Element ID	I
mtyp	Material ID of an element	I
iel	Number of nodes per element	I
lm(iel)	Node ID of an element	I
temp(iel)	Temperature of each node of an element	I
alpha(iel)	Seebeck coefficient of each node of an element	O
ne_nx(6)	Element IDs of elements that adjoin each element face of an element	I
mty_nx(6)	Material IDs of elements that adjoin each element face of an element.	I
node_f(4,6)	Internal node IDs of each element face of an element	I

5-29-7-3. Example

In this example, the Seebeck coefficient of each node is set to le-4 times.

```
      subroutine  usrsee ( ne, mtyp, iel, lm, temp, alpha,
ne_nx, mty_nx,
     node_f)
c
      implicitr real*8 (a-h, o-z)
c
      dimension  lm(iel)
      dimension  temp(iel), alpha(iel)
c
      do i = 1, iel
alpha(i) = 1.d-4
      end do
c
      return
```

5-30. User Subroutine (Charge Distribution Analysis: CH) (JMAG-Studio)

A charge distribution analysis is an electric field analysis coupled with motion/diffusion of particles.

The charge distribution analysis allows calculation "to track changes of the electric field when charged particles move affected by an electric field."

Motion/diffusion of a single particle (i) is calculated using the following continuum equation:

$$\sum_{j-1}^{N} C_{ij}\frac{\partial n_j}{\partial} + \nabla n_i v_i - D_i \nabla^2 n_i = F(n_1, n_2, \ldots, n_N, \nabla n_1, \nabla n_2, \ldots, \nabla n_3, v_1, v_2, \ldots, v_N) \quad \ldots(1)$$

where,

N : Number of particle types

ni : Density of particle i

Cij : Coefficient representing transition from particle j to particle i

vi : Velocity of particle i

Di : Diffusion coefficient of particle I

Fi : Function depending on particle number, their spatial differential, and velocity

This analysis has the following advantages.

- More than one type of particle can be analyzed.
- Transition from one particle to another can be treated.
- Time dependent boundary conditions can be defined.

MEMO Henceforth, the second term on the left hand side of equation (1) is referred to as the advection term, the third term on the left hand side is referred to as the diffusion term, and the term on the right hand side is referred to as the user term.

5-30-1. Types of User Subroutine

■ ***usrdns***

Used to set the initial number of particles and initial charge.

■ ***velusr***

Used to define the velocity of each particle.

■ ***rgtusr***

Used to define the user term.

■ ***dtusr***

Used to define the time increment.

■ ***setdns***

Used to change the derived number of particles and set the spatial charge.

■ ***bmuser***

Used to define the moving of charge caused by electric discharge.

■ ***usrgap***

Used to define the charge caused by the electric discharge of air gap.

- Fig.1 provides the calling location of each subroutine.
- *usrdns*, *velusr*, *rgtusr*, *dtuser*, and *setdns* in the flowchart represent user subroutines used in this module.

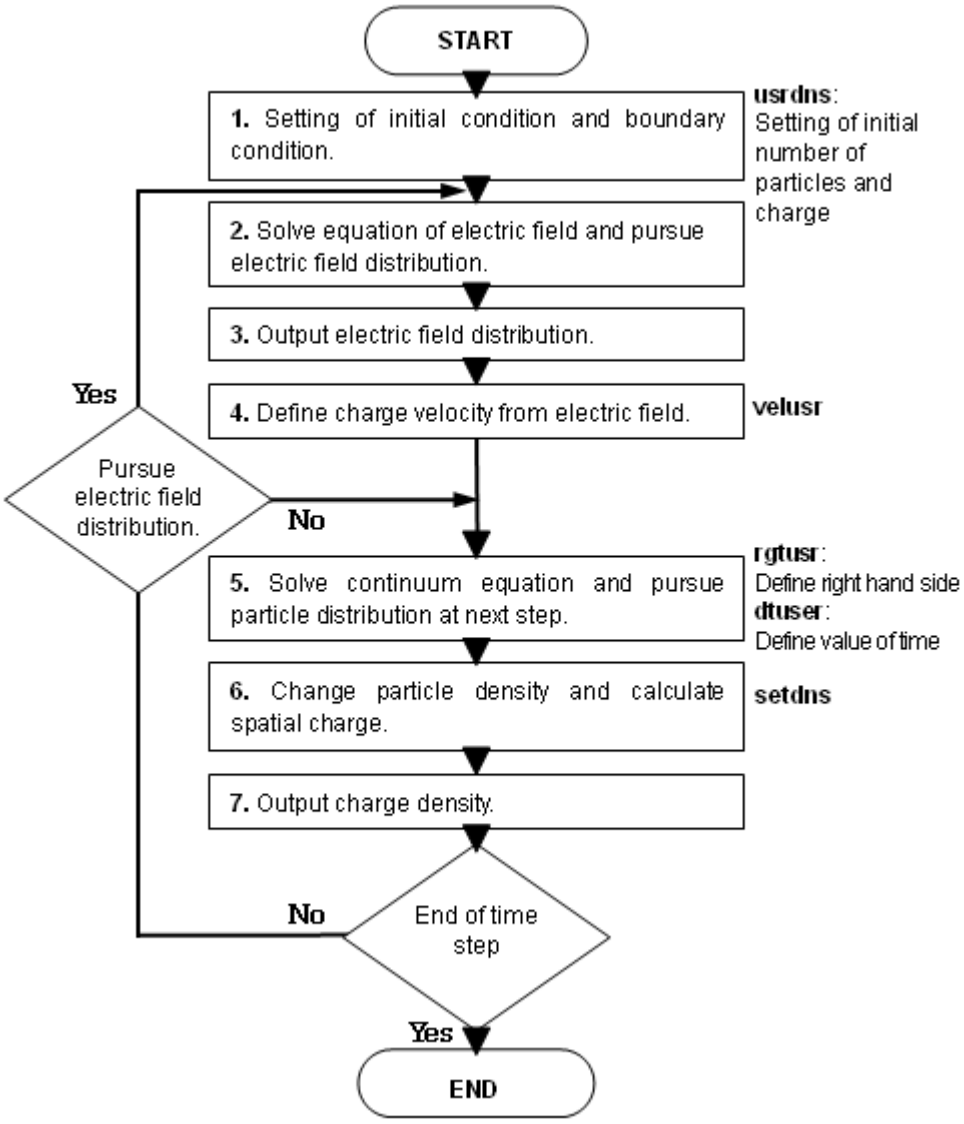

START
1. Setting of initial condition and boundary condition.
usrdns: Setting of initial number of particles and charge
2. Solve equation of electric field and pursue electric field distribution.
3. Output electric field distribution.
4. Define charge velocity from electric field.
velusr
Yes
Pursue electric field distribution.
No
5. Solve continuum equation and pursue particle distribution at next step.
rgtusr: Define right hand side
dtuser: Define value of time
6. Change particle density and calculate spatial charge.
setdns
7. Output charge density.
No
End of time step
Yes
END

Fig.1 Flowchart for a charge distribution analysis

5-30-2. Notes on Using User Subroutines

- Describe user subroutines using FORTRAN. JMAG-Studio is compatible with VisualStudio2005 and Intel Visual Fortran 9.1.
  - User subroutines for Intel Visual Fortran are found at \Solver\mod\elec\usrsub under the directory where JMAG-Studio is installed.
    Also use Intel Visual Fortran 9.1.
- The following device numbers are reserved. Use unused device numbers for user subroutines.

  Reserved device numbers: 1, 2, 6, 10, 11, 20, 21, 22, 23, 24, 55
- The electric potential, electric field, and particle density of each particle are exported as the analysis results.

5-30-3. Reconfiguration of JMAG-Studio

- To use user subroutines, after Fortran program file (*.f) is rewritten arbitrarily, the total module needs to be reconfigured.

5-30-3-1. JMAG-Studio (Intel Visual Fortran 9.1)

1. **Make sure that VisualStudio2005 is installed to the machine.**

   In case it is not installed, a module cannot be created.
2. **Open the Fortran program file (*.f) placed in the following directory, rewrite the contents, and save.**

   `(JMAG installation directory)\solver\mod\elec\usrsub`

   Do not change the directory and file name of the Fortran program file.
   It is recommend to create a backup file for the Fortran program file prior to rewriting.
3. **Start Up VisualStudio2005.**
4. **Open usrsub.sln.**

   usrsub.sln is located in the same directory in step 2.
   Select [File] > [Open] > [Project] from the menu bar, and then open usrsub.sln.

5. When right-clicking [Project] of [Solution Explorer], if [Extract Compaq visual Fortran Project Items] is displayed in the shortcut menu, select it, and then specify usrsub.dsp.

usrsub.dsp is also located in the same directory in step 2. [Extract Compaq Visual Fortran Project Items] is displayed in the menu for the first time of compiling the program with your machine.

If [Extract Compaq Visual Fortran Project Items] is not displayed in the menu, proceed to step 6.

6. Execute a reconfiguration of the module (elec.exe).

Select [Build] > [Build of solution] or [Build] > [Rebuild of solution] from the menu bar, and execute the reconfiguration.

When the reconfiguration is executed, a new module (elec.exe) is created in the following directory.

```
(JMAG installation directory)\solver\mod\elec\mod
```

7. Place the module (elec.exe) created in step 6 in the installation directory of JMAG-Studio.

At this time, it is recommend to place a different name on the original elec.exe and save it.

8. Execute an analysis with JMAG-Studio.

Important **When creating a user subroutine for 64 bit using VisualStudio2005, perform the following steps to set up the development environment.**

1. Open the project with VisualStudio2005.

2. Click the Solution Platform list (labeled as Win32), and if available, select [x64] to enable the build configuration.

If "64 bit" does not appear in the list, add it to the list as follows.

3. Select [Configuration Manager] from the [Solution Platform] list.

4. Select [New] from the [Active Solution Platform] list.

5. Select [x64] from the [Enter or Select New Platform] list.

6. [x64] can be selected from the [Solution Platform] list in step 2.

Important **The build environment for Intel 64 needs to be added by selecting [Custom Install] when installing Visual Studio.**

Important **When installing Visual Studio and Intel Fortran on the same machine, make sure that they are installed in the correct order (first Visual Studio, and then Intel Fortran), and they are set up for building the 64-bit module. For more information, see the manual of each toolkit.**

5-30-3-2. Reconfiguration of the JMAG-Studio Linux Module

1. **Place the Fortran program file (*.f) compiled with a procedure similar to the aforementioned “Reconfiguration of JMAG-Studio” in the following directory.**

   `(JMAG installation directory)solver\mod\elec\usrsub`

2. **Replace it under the directory of step 1.**

3. **Execute [%make] to start a reconfiguration.**

   Here “%” means the prompt.

5-30-4. usrdns

5-30-4-1. Settings

- *usrdns* defines the initial number of particles and initial spatial charge.
- If charge distribution analysis data (NDNS0) is 0, always set this subroutine.

5-30-4-2. Arguments

Variable	Details	I/O
dnsty(numnp,npart)	Number of particles on each node	O
charg(numnp)	Spatial charge on each node	O
charg0(numnp)	Spatial charge on each node	O
coord(3,numnp)	Coordinate of each node	I
volum(numnp)	Volume of each node	I
numnp	Total number of nodes	I
npart	Number of particle types	I
codorg(3, numnp)	Coordinate of node at the initial position	I

5-30-4-3. Example

In this example, particle density (1.0d+3) is set for nodes with z-coordinate greater than (5.0d-4).

■ Program

```
      subroutine usrdns(dnsty,charg,charg0,
     #             coord,volum,numnp,npart)
      implicit real*8 (a-h,o-z)
c
      dimension dnsty(numnp,npart),charg(numnp),charg0(numnp),
     #        coord(3,numnp),volum(numnp)
c
      common /file/ ifl01,ifl05,ifl06,
     #           ifl10,ifl11,ifl12,ifl13,ifl14,ifl15,
     #           ifl20,ifl21,ifl30,ifl31,
     #           ifl40,ifl41,ifl42
c
      dl  = 1.0d-7
      roo = 1.d+3
c
      do 100 np=1,numnp
         if (coord(3,np).gt.5.00d-4-dl) then
            dnsty(np,1) = roo*volum(np)
         endif
100 continue
c
     return
     end
```

5-30-5. velusr

5-30-5-1. Settings

- *velusr* defines the advection velocity of each particle
- This subroutine must be defined.

5-30-5-2. Arguments

Variable	Details	I/O
velnp(3,numnp,npart)	Advection velocity vector on each node	O
vlabs(numnp,npart)	Absolute value of advection velocity on each node	I
evcnp(3,numnp)	Electric field vector on each node	I
evabs(numnp)	Absolute value of electric field vector on each node	I
alpha(numnp)	Usable in *rgtusr*	O
dnsty(numnp, npart)	Number of particles on each node	I
volum(numnp)	Volume on each node	I
coord(3,numnp)	Coordinate of each node	I
numnp	Total number of nodes	I
npart	Number of particle types	I
nt	Current time step number	I
toimpt	Current time	I
codorg(3, numnp)	Coordinate of a node at the initial position	I

5-30-5-3. Example

In this example, velocities for two particle are defined as a function of electric field vector.

- For particle 1, the velocity is 10 times larger than the electric field vector and has the same direction as the electric field.
- For particle 2, the velocity is 100 times larger than the electric field vector and has the opposite direction of the electric field.

■ Program

```
      subroutine velusr(velnp,vlabs,
     #             evcnp,evabs,alpha,
     #             dnsty,volum,coord,
     #             numnp,npart,nt,timopt)
      implicit real*8 (a-h,o-z)
c
      dimension velnp(3,numnp,npart),vlabs(numnp,npart),
     #        evcnp(3,numnp),evabs(numnp),alpha(numnp),
     #        dnsty(numnp,npart),volum(numnp),coord(3,numnp)
c
      common /file/ ifl01,ifl05,ifl06,
     #           ifl10,ifl11,ifl12,ifl13,ifl14,ifl15,
     #           ifl20,ifl21,ifl30,ifl31,ifl40,ifl41,ifl42
c
      coef1 = 100.d0
      coef2 = -10.d0
c
      do 100 np=1,numnp
         velnp(1,np,1) = coef1*evcnp(1,np)
         velnp(2,np,1) = coef1*evcnp(2,np)
         velnp(3,np,1) = coef1*evcnp(3,np)
c
         velnp(1,np,2) = coef2*evcnp(1,np)
         velnp(2,np,2) = coef2*evcnp(2,np)
         velnp(3,np,2) = coef2*evcnp(3,np)
  100 continue
c
      return
      end
```

5-30-6. rgtusr

5-30-6-1. Settings

- *rgtusr* defines a user term in the equation (1).
- To solve the equation containing a user term, this subroutine must be defined.

5-30-6-2. Arguments

Variable	Details	I/O
right(numnp,npart)	Value of the user term on each node	O
dnsty(numnp,npart)	Number of particles on each node	I
difsn(numnp,npart)	Value of the diffusion term on each node	I
evcnp(3,numnp)	Electric field vector on each node	I
evabs(numnp)	Absolute value of electric field vector on each node	I
alpha(numnp)	Value defined in velset	I
velnp(3,numnp,npart)	Advection velocity vector on each node	I
vlabs(numnp,npart)	Absolute value of advection velocity on each node	I
coord(3,numnp)	Coordinate of each node	I
volum(numnp)	Volume of each node	I
numnp	Total number of nodes	I
npart	Number of particle types	I
nt	Current time step number	I
timopt	Current time	I
dtopt	Time increment	I
codorg(3, numnp)	Coordinate of a node at the initial position	I

5-30-6-3. Example

In this example, the user term of (number of particles) x (absolute value of particle velocity) is defined for each node.

■ **Program**

```
      subroutine rgtusr(right,
     #             dnsty,difsn,
     #             evcnp,evabs,alpha,
     #             velnp,vlabs,
     #             coord,volum,
     #             numnp,npart,nt,timopt,dtopt)
      implicit real*8 (a-h,o-z)
c
      dimension right(numnp,npart),
     #          dnsty(numnp,npart),difsn(numnp,npart),
     #          evcnp(3,numnp),evabs(numnp),alpha(numnp),
     #          velnp(3,numnp,npart),vlabs(numnp,npart),
     #          coord(3,numnp),volum(numnp)
c
      common /file/ ifl01,ifl05,ifl06,
     #           ifl10,ifl11,ifl12,ifl13,ifl14,ifl15,
     #           ifl20,ifl21,ifl30,ifl31,
     #           ifl40,ifl41,ifl42
c
c
c
      do 100 np=1,numnp
          right(np,1) = dnsty(np,1)*vlabs(np,1)
                right(np,2) = dnsty(np,2)*vlabs(np,2)
  100 continue
c
c
c
       return
       end
```

5-30-7. dtuser

5-30-7-1. Settings

- *dtuser* defines time increment.
- If charge distribution analysis data (IDTUSR) is 1, the time increment must be set.

5-30-7-2. Arguments

Variable	Details	I/O
velmax(npart)	Maximum value of the advection velocity of each node	I
npart	Number of particle types	I
vmax	Maximum particle velocity	I
dxmin	Minimum value of mesh interval	I
dtopt	Time increment	O
timopt	Current time	I
dnsty(numnp,npart)	Number of particles on each node	I
numnp	Total number of nodes	I
iprnt2	Output flag	I
icalc	Electric field analysis flag	I

5-30-7-3. Example

In this example, the time increment is defined to be one-tenth of (minimum value of mesh interval) / (maximum particle velocity).

■ **Program**

```
      subroutine dtuser(velmax,npart,vmax,dxmin,dtopt,timopt)
      implicit real*8 (a-h,o-z)
c
      dimension velmax(npart)
c
      common /file/ ifl01,ifl05,ifl06,
     #           ifl10,ifl11,ifl12,ifl13,ifl14,ifl15,
     #           ifl20,ifl21,ifl30,ifl31,
     #           ifl40,ifl41,ifl42
c
c
c
      dtopt = 0.1*dxmin/vmax
c
c
c
      return
      end
```

5-30-8. setdns

5-30-8-1. Settings

- *setdns* changes the number of particles and calculates the spatial charge.
- The spatial charge calculated here is taken into account in the electric field analysis. This subroutine should be set if necessary.

5-30-8-2. Arguments

Variable	Details	I/O
dnsty(numnp,npart)	Number of particles on each node	O
charg(numnp)	Spatial charge on each node	I
charg0(numnp)	Initial spatial charge on each node	I
evcnp(3,numnp)	Electric field vector on each node	I
evabs(numnp)	Absolute value of electric field vector on each node	I
velnp(3,numnp,npart)	Advection velocity vector on each node	I
vlabs(numnp,npart)	Absolute value of advection velocity on each node	I
coord(3,numnp)	Coordinate of each node	I
volum(numnp)	Volume on each node	I
numnp	Total number of nodes	I
npart	Number of particle types	I
nt	Current time step number	I
timopt	Current time	I
codorg(3, numnp)	Coordinate of a node at the initial position	I
fai	Electric potential	I

5-30-8-3. Example

In this example, the spatial charge of (number of particles) x (charge of electron) is defined.

■ **Program**

```
      subroutine setdns(dnsty,charg,charg0,
     #              evcnp,evabs,
     #              velnp,vlabs,
     #              coord,volum,
     #              numnp,npart,nt,timopt)
      implicit real*8 (a-h,o-z)
c
      dimension dnsty(numnp,npart),charg(numnp),charg0(numnp),
     #        evcnp(3,numnp),evabs(numnp),
     #        velnp(3,numnp,npart),vlabs(numnp,npart),
     #        coord(3,numnp),volum(numnp)
c
      common /file/ ifl01,ifl05,ifl06,
     #           ifl10,ifl11,ifl12,ifl13,ifl14,ifl15,
     #           ifl20,ifl21,ifl30,ifl31,
     #           ifl40,ifl41,ifl42
c
c
c
      elect = 1.602d-19
c
      do 100 np=1,numnp
         cahrg(np) = charg0(np) - elect*dnsty(np,1)
 100  continue
c
c
c
      return
      end
```

5-30-9. bmuser

5-30-9-1. Settings

- *bmuser* defines the moving of charge caused by electric discharge.
- The charge between two nodes that are connected with a beam element is defined.

5-30-9-2. Arguments

Variable	Details	I/O
ni	Particle type	I
nowelm	Beam element ID	I
np1	First node ID of beam element	I
np2	Second node ID of beam element	I
fai1	Electric potential of the first node of beam element	I
fai2	Electric potential of the second node of beam element	I
dist	Length of beam element	I
velmp(3,numnp,npart)	Advection velocity vector on each node	I
evcnp(3,numnp)	Electric field vector on each node	I
evabs(numnp)	Absolute value of electric field vector on each node	I
dnsty(numnp,npart)	Number of particles on each node	O
volum(numnp)	Volume on each node	I
coord(3,numnp)	Coordinate of each node	I
numnp	Total number of nodes	I
npart	Number of particle types	I
nt	Current time step number	I
timopt	Current time	I
np1flg	First node control flag	O
np2flg	Second node control flag	O
dnsnp1	Number of particles moving to the first node	O
dnsnp2	Number of particles moving to the second node	O
codorg(3,numnp)	Original node coordinate	O

5-30-9-3. Example

■ Program

```
      subroutine bmuser(ni,nowelm,
     #                  np1,np2,fai1,fai2,dist,
     #                  velnp,evcnp,evabs,dnsty,
     #                  volum,coord,
     #                  numnp,npart,nt,timopt,
     #                  np1flg,np2flg,dnsnp1,dnsnp2, codorg)
      implicit real*8 (a-h,o-z)
c
      dimension velnp(3,numnp),
     #          evcnp(3,numnp),evabs(numnp),
     #          dnsty(numnp,npart),
     #          volum(numnp),coord(3,numnp), odorg(3,numnp)
c
      common /file/ ifl01,ifl05,ifl06,
     #           ifl10,ifl11,ifl12,ifl13,ifl14,ifl15,
     #           ifl20,ifl21,ifl30,ifl31,
     #           ifl40,ifl41,ifl42
c
c
c
      np1flg = 1
c
c
c
      return
      end
```

5-30-10. usrgap

5-30-10-1. Settings

- *usrgap* defines the moving of charge caused by the electric discharge of air gap.

5-30-10-2. Arguments

Variable	Details	I/O
time	Current time	I
ntyp	Particle type	I
ng	Gap ID	I
ne (2)	Element ID of gap	I
mt (2)	Material ID of gap element	I
nd (2)	Node ID of gap	I
v (2)	Electric potential of nodes of gap	I
dns (2)	Number of particles of gap	I/O
chg (2)	Charge of gap	I
eps (2)	Relative permittivity	I
g	Gap length	I
eps00	Vacuum permittivity	I
lspark	Electric discharge flag	O

5-30-10-3. Example

In this example, the electric charge exceeds the discharged potential difference is set according to the Paschen's law.

■ **Program**

```
      subroutine  usrgap(time,ntyp,ng,ne,mt,nd,
     #                   V,dns,chg, eps , g , eps0 , lspark)
      implicitreal*8 (a-h,o-z)
c
      real*8   time
      integer  ntyp
      integer  ng, ne(2), mt(2), nd(2)
      real*8       V(2), dns(2), chg(2), eps(2), g, eps0
      integer      lspark
c
      real*8       q
      parameter  (q = 1.0d-10)
      real*8       Vpa, gum
      integer      high, low
      real*8       dV
      real*8       V1, V2
      integer      mt1, mt2
      real*8       d1, d2, D, ep1, ep2, ep0, dq, dn
      real*8       zero
      parameter   (zero = 0.0d0)
c
       lspark = 0
c
      if (time .lt. 1.0d-7)                                  return
c
      if (ntyp .eq. 2)                                       return
c
      if (time .gt. 9d-3) then
        gum = g * 1.0d+6
      end if
      if (time .gt. 1d-2) then
        gum = g * 1.0d+6
      end if
      if (time .gt. 1.6d-2) then
        gum = g * 1.0d+6
      end if
      if (time .gt. 2d-2) then
        gum = g * 1.0d+6
      end if
c
c     gap distance in micro-meter
        gum = g * 1.0d+6
c
c     Paschen's law
      if      (gum .lt. 4.800d0) then
```

```
        Vpa = 75.d0 * gum
      else if (gum .lt. 7.742d0) then
       Vpa = 360d0
      else
      Vpa = 312d0 + 6.2d0 * gum
      end if
c
     write(*,*) ' *** usrgap Vpa -> 1/2 for test execution ***'
        Vpa = Vpa / 2
c
c     check high voltage low voltage
      if (V(1) .gt. V(2)) then
                                  high = 1
                                  low  = 2
      else
           high = 2
          low  = 1
      end if
c
c     voltage difference
       dV = V(high) - V(low)
c
c     charge amount
      if      (ntyp .eq. 1) then
        chg(high) =  q * dns(high)
        chg(low ) =  q * dns(low)
       if  (chg (high) .le. 0.0d0)
return
      else if (ntyp .eq. 2) then
        chg (low ( = -q * dns)low)
       if (chg(low ) .ge. 0.0d0)                          return
      else
       write(*,*) '*** illegal type ***', ntyp

return
      end if
c
c     not discharge
     if (dV .lt. Vpa)                                    return
c
c     set high voltage side to 1, low voltage side to 2
       ep0 = eps0
       ep1 = eps(high)
       ep2 = eps(low)
c     thickness must be set using material-ID, etc.
       mt1 = mt(high)
       mt2 = mt(low)
c       d1, d2 in unit micro-meter)
        d1 = 0.1d+3
        d2 = 0.1d+3
c
       if (ntyp .eq. 1) then
        if (mt1 .eq. 2  .and.  mt2. eq. 1
```

```
     .                    .or.
     .       mt1 .eq. 3   .and.  mt2. eq. 2) then
        else
                                                              return
        end if
       else if (ntyp .eq. 2) then
                                                              return
       end if
c
        D  =  d1 / ep1  +  d2 / ep2
c
        V1 = dV
        V2 = Vpa
c
      if (gum .eq. zero) then
       if      (ntyp .eq. 1) then
        dq = chg(high)
       else if (ntyp .eq. 2) then
        dq = chg(low)
       end if
c
      else
c
c      discharged charge
        dq = (V1 - V2) * (D + gum) / (D * gum) * ep0
c        unit um -> m  and Area
        dq = dq * 1.0d6 * (1.0d-4)**2
        dq = dq * 5.65d-5
        dq = dq * 1.0d-2
       if (ntyp .eq. 2) dq = -dq
c
c      limit of charge
      if ( ntyp.eq. 1  .and.  dq .gt. chg(high)) dq = chg(high)
       if (ntyp.eq. 2  .and.  dq .lt. chg(low)) dq = chg(low)
c
      end if
c
c     no discharged charge
      if (abs( dq ) .lt. 1.0d-30 )                          return
c
       if   ( ntyp .eq. 1 ) then
        chg(high) = chg(high) - dq
        chg(low ) = chg(low ) + dq
        dns(high) = dns(high) - dq / q
        dns(low ) = dns(low ) + dq / q
       else if ( ntyp .eq. 2 ) then
        chg(high) = chg(high) + dq
        chg(low ) = chg(low ) - dq
        dns(high) = dns(high) - dq / q
        dns(low ) = dns(low ) + dq / q
       end if
c
       lspark = 1
```

```
c
      write(1,'(a,i3,f10.3,i5,i2,1p,4e10.3)')
     .  ' Paschen spark ', ntyp, gum, mt1, mt2, dV, Vpa, time,
dq
c
      end
```

5-31. Output Items of Electric Field Analysis

The electric field analysis module (EL, CH) can display the following items as analysis results.

MEMO Each of the export items are displayed in units that are applied to an analysis model in JMAG-Designer.

Output item	Unit	Type	Object	Note
Electric potential	V	scalar	node	
Electric field	V/m	vector	solid	
Force by electric field	N	step	-	This is exported when the force calculation condition is set in JMAG-Studio or a force condition is set in JMAG-Designer for the static analysis and frequency response analysis.
Current	A	step	-	This is exported when the current calculation condition is set in JMAG-Studio or the current condition is set in JMAG-Designer for the current distribution analysis and the frequency response analysis.
Current density	A/m^2	scalar	shell, solid	This is exported for the current distribution analysis and frequency response analysis.
Power consumption	W	step	-	This is exported when power consumption is selected for a current distribution analysis.
Power consumption density	W/m^3	scalar	solid	This is exported when power consumption is selected for a current distribution analysis.
Thermal electromotive force loss	W	step	-	This is exported when thermal electromotive force loss is selected for a current distribution analysis when coupling to a thermal analysis using the Seebeck coefficient.

Output item	Unit	Type	Object	Note
Thermal electromotive force loss density	W/m^3	scalar	solid	This is exported when thermal electromotive force loss is selected for a current distribution analysis when coupling to a thermal analysis using the Seebeck coefficient.
Total loss	W	step	-	This is exported when total loss is selected for a current distribution analysis when coupling to a thermal analysis using the Seebeck coefficient.
Total loss density	W/m^3	scalar	solid	This is exported when total loss is selected for a current distribution analysis when coupling to a thermal analysis using the Seebeck coefficient.
Electric charge	C	step	-	This is exported when the surface charge distribution calculation condition is set in JMAG-Studio or the surface charge condition is set in JMAG-Designer for the static analysis and frequency response analysis.
Amount of charge	C	scalar	node	This is exported for the static analysis and frequency response analysis. For the charge distribution analysis, this is exported if charge is set at *setdns* of the user subroutine *oluser*f.
Surface charge	C/m^2	scalar	shell	This is exported when the surface charge distribution calculation condition is set for the static analysis and frequency response analysis.

Output item	Unit	Type	Object	Note
Nodal charge	C	scalar	node	This is exported when internal charge is selected and there is a nodal charge for the static analysis and frequency response analysis.
Line charge density	C/m	scalar	beam	This is exported when internal charge is selected and there is a line charge for the static analysis and frequency response analysis.
Face charge density	C/m^2	scalar	shell	This is exported when internal charge is selected and there is a surface charge for the static analysis and frequency response analysis.
Particle number density	Pcs/m^3	scalar	node	This is exported for the charge distribution analysis.
Nodal force	N	scalar	node	This is exported when the force calculation condition is set in JMAG-Studio or a force condition is set in JMAG-Designer for the static analysis and frequency response analysis.
Polarization vector	C/m^2	scalar	solid	This is exported when internal charge is selected and there is a polarization vector for the static analysis and frequency response analysis.

MEMO Difference between a surface charge and an internal charge:
A surface charge is exported when the surface charge distribution calculation condition is set. The surface charge distribution and the amount of surface charge on the face where the surface charge distribution calculation condition is set are exported. To calculate the capacity of a capacitor, use the surface charge obtained.
An internal charge (node charge, line charge density, face charge density, and polarization vector) is exported when the [Internal Charge] check box is selected for the analysis control condition for JMAG-Studio and the study properties for JMAG-Designer. If the integral method calculator tool (CALCB) is used to calculate the magnetic field at the specified position in the air, the internal charge needs to be exported to the plot file of the magnetic field analysis to reference.

Chapter 6. Function of Structural Analysis (JMAG-Designer)

This chapter describes in detail the material properties and conditions used in structural analyses.

▶▶ Reference

For information about setting materials and conditions, see "JMAG-Designer User's Manual."

6-1. Overview of Functions

6-1-1. DS (Structural Analysis Module)

The structural analysis module (DS) is used for the static, eigenmode, and frequency response analyses.

■ Static analysis

- A static analysis is used to evaluate displacement and stress when static load acts on a structure.
- The temperature load can be set.

■ Eigenmode analysis

- An eigenmode analysis is used to evaluate the eigen value and eigen vector of a structure.

■ Frequency response analysis

- A frequency response analysis is used to evaluate the response of a structure that constantly vibrates.
- Displacement, velocity, acceleration, and stress at an arbitrary frequency can be obtained as calculation results.
- In combination with a magnetic field analysis (transient response analysis), electromagnetic force can be taken into account in an analysis (one way coupled analysis).

- In combination with a magnetic field analysis (frequency response analysis), electromagnetic force can be taken into account in an analysis (one way coupled analysis).

  ▶▶ Reference

  For more information about coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

6-2. Conditions of Structural Analysis (JMAG-Designer)

JMAG-Designer supports the following analysis condition. The available functions vary depending on the analysis type.

Analysis Condition	2D			3D		
	DS-ST	DS-EIG	DS-FQ	DS-ST	DS-EIG	DS-FQ
Material Database						
Material (isotropic)	O	O	O	O	O	O
Material (anisotropic)	O	O	O	O	O	O
Part properties						
Beam	O	O	O	O	O	O
Shell	O	O	O	O	O	O
Solid	O	O	O	O	O	O
Boundaries						
Electric potential boundary	X	X	X	O	O	O
Load						
Concentrated load	O	X	O	O	X	O
Pressure load	O	X	O	O	X	O
Acceleration load	O	X	X	O	X	X
Volume load	X	X	X	O	X	O
Centrifugal Force	O	X	X	O	X	X
Electromagnetic force	O	X	O	O	X	O
Temperature load	O	X	X	O	X	X
Press fit	O	X	X	X	X	X
Constraint						
Displacement	O	O	O	O	O	O
Rigid body	O	O	O	O	O	O
Constraint	O	O	O	O	O	O
Rotation periodic boundary	O	O	O	O	O	O
Contact						
Adhesion	O	X	X	X	X	X

Analysis Condition	2D			3D		
	DS-ST	DS-EIG	DS-FQ	DS-ST	DS-EIG	DS-FQ
Spring-Mass						
Spring	O	O	O	O	O	O
Concentrated mass	X	O	O	X	O	O
Output						
Sound pressure	X	X	O	X	X	O
Study Properties						
Solver calculation control	O	O	O	O	O	O
Eigen value	X	⊙	⊙	X	⊙	⊙
Frequency control	X	X	⊙	X	X	⊙
Output control	O	X	O	O	X	O

⊙: Required O: Optional X: Not available

- DS-ST: Structural analysis (static analysis) *3D, 2D
- DS-EIG: Structural analysis (eigenmode analysis) *3D, 2D
- DS-FQ: Structural analysis (frequency analysis) *3D, 2D

6-3. Material Property for Each Part: Beam

▶▶ Reference

For more information, see Chapter 28, "Assigning Materials and Setting Material Properties" in the JMAG-Designer User's Manual.

6-3-1. Function

The beam properties can be specified.

■ Related items

- Material: Mechanical Properties

■ Settings

- Type of the cross section
- Area of the cross section
  - Radius of a circle (m), (cm), (mm), (μm), (nm), (mil), (ft), (in)
  - Edge length of a square (m), (cm), (mm), (μm), (nm), (mil), (ft), (in)
  - Length and width of a rectangular (m), (cm), (mm), (μm), (nm), (mil), (ft), (in)

■ Available analyses

- All types of structural analyses

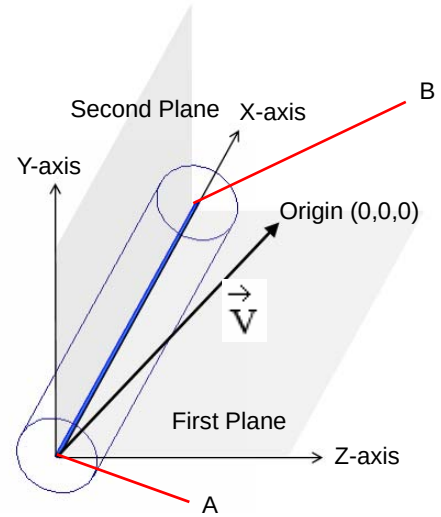
■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-3-2. Important Notes Regarding the Use of This Function

- The degree of freedom for beam elements is 6 (translational freedom is 3, and rotational freedom is 3). However, The degree of rotational freedom cannot be specified for a constraint condition and displacement condition because the rotational freedom is automatically determined in JMAG-Designer.
- A beam element can be defined as follows.

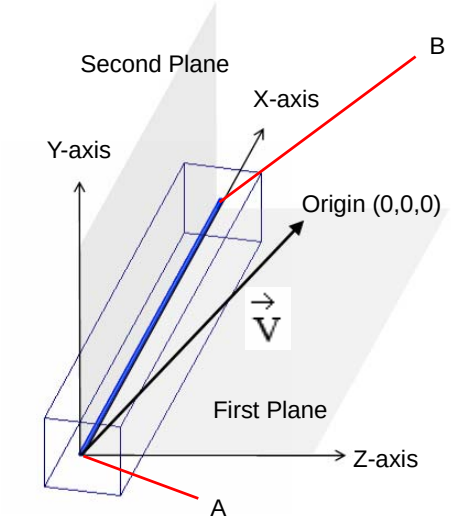
■ Circle Cross Section



- $\vec{V}$: A vector comprised of vertex A and the origin
- X-axis: An axis passing through vertex A and vertex B.
- Y-axis: An axis perpendicular to the X-axis and positioned on the first plane.
- Z-axis: An axis passing through vertex A and perpendicular to the X and Y axes
- First plane: A plane comprised of the X and $\vec{V}$ axes

- The beam element properties are defined based on the figure above.
  - Section Secondary Moment 1: A moment about the Z-axis
  - Section Secondary Moment 2: A moment about the Y-axis
  - Torsional Constant: A moment about the X-axis

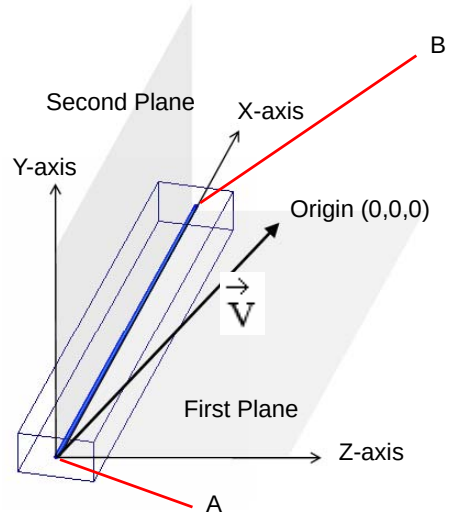
■ Square Cross Section



- $\vec{V}$: A vector comprised of vertex A and the origin
- X-axis: An axis passing through vertex A and vertex B.
- Y-axis: An axis perpendicular to the X-axis and positioned on the first plane.
- Z-axis: An axis passing through vertex A and perpendicular to the X- and Y axes
- First plane: A plane comprised of the X and $\vec{V}$ axes

- The beam properties are defined based on the figure above.
  - Section Secondary Moment 1: A moment about the Z-axis
  - Section Secondary Moment 2: A moment about the Y-axis
  - Torsional Constant: A moment about the X-axis

■ Rectangular Cross Section



- $\vec{V}$: A vector comprised of vertex A and the origin
- X-axis: An axis passing through vertex A and vertex B.
- Y-axis:An axis perpendicular to the X-axis positioned on the first plane.
- Z-axis: An axis passing through vertex A and perpendicular to the X and Y axes
- First plane: A plane comprised of the X and $\vec{V}$ axes
- Secondary plane: A plane parallel to the X and Z axes

- The beam element properties are defined based on the figure above.
  - Section Secondary Moment 1: A moment about the Z-axis
  - Section Secondary Moment 2: A moment about the Y-axis
  - Torsional Constant: A moment about the X-axis

6-4. Material Property for Each Part: Shell

▶▶ Reference

For more information, see Chapter 28, "Assigning Materials and Setting Material Properties" in the JMAG-Designer User's Manual.

6-4-1. Function

The shell properties can be specified.

■ Related items

- Material: Mechanical Properties

■ Settings

- Thickness of a shell(m), (cm), (mm), (μm), (nm), (ft), (mil), (in)
- Bending stiffness parameter
- Ratio of transverse shear thickness to membrane
- Distance from the neutral plane (top face, bottom face) (m), (cm), (mm), (μm), (nm), (ft), (mil), (in)

■ Available analyses

- All types of structural analyses

■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

6-4-2. Important Notes Regarding the Use of This Function

- The degree of freedom for shell elements is 5 (translational freedom is 3, and rotational freedom is 2 (excluding the rotation about the normal line on the element face)). However, The degree of rotational freedom cannot be specified for a constraint condition and displacement condition because the rotational freedom is automatically determined in JMAG-Designer.
- The thickness of a shell is used to map electromagnetic force accounting for the thickness when both the model for the magnetic field analysis results referred to from the electromagnetic force condition and the model for the structural analysis are 2D.

▶▶ Reference

For more information, see P.801 "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction".

6-5. Material Property for Each Part: Solid

6-5-1. Function

The solid properties can be specified.

■ Related items

- Material: Mechanical Properties

■ Settings

- Coordinate system (Anisotropic material)

■ Available analyses

- All types of structural analyses

6-5-2. Important Notes Regarding the Use of This Function

- The degree of freedom for solid elements is 3 (translational freedom is 3, and rotational freedom is 0).

6-6. Electric Potential Boundary (Piezoelectric analysis)

▶▶ Reference
See Chapter 32. "Study Properties and Conditions of Structural Analysis", "Electric Potential Boundary" of the JMAG-Designer User's Manual.

6-6-1. Function

Specifies electrode and electric potential when running a piezoelectric analysis.

The piezoelectric analysis is a function to run an analysis using the effects of structural and electric field. This is done by setting the piezoelectric stress constant and permittivity settings that is defined in a material (hereafter piezoelectric material) and set in a part. An analysis for each type of sensor using the piezoelectric result can be done by adding force to the part set with piezoelectric material. An analysis for each type of actuator can be run by applying voltage to parts set with piezoelectric material.

■ Related items

- Materials with electric properties (Relative permittivity)
- Materials with mechanical properties (Piezo electric stress constant)

■ Setting

- Static analysis: Electric potential (amplitude)
- Frequency response analysis: Electric potential (amplitude, phase)

■ Analysis that can be used with this function

- Entire 3D analysis

■ Output item

- Electric Potential (V)

6-6-2. Important Notes Regarding This Function

- Specify the electric potential to '0V' when the eigenmode analysis is set.
- Specify piezoelectric material that differs for each polar plate when running a piezoelectric analysis.
- An analysis is run using the largest value in electric potential that is set in an area, when electric potential that are overlapping is set in a region.

6-7. Concentrated Load

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis", "Concentrated Load" of the JMAG-Designer User's Manual.

6-7-1. Function

The concentrated load condition is used to specify a load for a vertex.

■ Related items

- None

■ Settings

- Load for vertex
- Load (N) or (kgf)
- Phase (deg) or (rad)
- Direction of the load

■ Available analyses

- Static analysis
- Frequency analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)
- Nodal force (input load) (N)* The output of nodal force is only available for 3D analyses.

6-8. Pressure Load

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Pressure Load” of the JMAG-Designer User's Manual.

6-8-1. Function

The pressure load condition is used to specify a pressure to be applied to a face or an edge.

■ Related items

- None

■ Settings

- Load acting on face or edge
- Load(Pa)
- Phase (deg) or (rad)
- Direction of the load

■ Available analyses

- Static analysis
- Frequency analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Nodal force (input load) (N)* The output of nodal force is only available for 3D analyses.

6-9. Volume Load

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Volume Load" of the JMAG-Designer User's Manual.

6-9-1. Function

The volume load condition is used to specify the total load acting on an entire part.

■ Related items

- None

■ Settings

- Load acting on part
- Amount of load (N) or (kgf)
- Phase (deg) or (rad) * For frequency analyses
- Direction of the load

■ Available analyses

- Static analysis
- Frequency analysis

■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)
- Nodal force (input load)(N)* The output of nodal force is only available for 3D analyses.

6-9-2. Important Notes Regarding the Use of This Function

- This condition cannot be used for shell elements.

6-10. Acceleration Load

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Acceleration Load” of the JMAG-Designer User's Manual.

6-10-1. Function

The acceleration load condition is used to specify acceleration acting on parts.

■ Related items

- None

■ Settings

- Load acting on part
- Acceleration speed(m/sec^2)
- Direction of the load

■ Available analyses

- Static analysis

■ Output items

- Nodal displacement(m)
- Stress(Pa)
- Electric potential (V)
- Charge density(C/m^3)
- Nodal force (input load) (N)

6-11. Centrifugal Force

Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Centrifugal Force" of the JMAG-Designer User's Manual.

6-11-1. Function

The centrifugal force condition specifies centrifugal force that acts on a rotating body in static analysis.

■ Related items

- None

■ Settings

- Load for part
- Rotation velocity(rpm) or (rps)
- Axis

■ Available analyses

- Static analysis

■ Output items

- Nodal displacement(m)
- Stress (Pa)
- Nodal force (input load)(N)

6-11-2. Important Notes Regarding the Use of This Function

- The centrifugal force condition cannot be set multiple times.
- The rotation axis can be specified.

6-12. Electromagnetic Force

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Electromagnetic Force” of the JMAG-Designer User's Manual.

6-12-1. Function

The electromagnetic force condition is used to specify a load for a structural analysis using the electromagnetic force obtained from a magnetic field analysis (one way coupled analysis).

If a result file of frequency analysis (magnetic field analysis) is referenced, a load is specified for a structural analysis using nodal force.

■ Related items

- Shell
- Coupled structural and magnetic field analysis

■ Settings

- Magnetic field analysis result file (Jplot or plot file) to reference
- Magnetic field analysis input file (JCF file) to reference

  * Only when a plot file is referenced.
- Step and frequency to reference
- FFT resolution

  * When the structural analysis (frequency analysis) and the magnetic field analysis (transient analysis) are used.
- Coordinate system to reference

  * When the structural analysis uses a 3D model and the magnetic field analysis uses a 2D model.

■ Available analyses

- Static analysis
- Frequency analysis

■ Output items

- Frequency (Hz)

- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-12-2. Important Notes Regarding the Use of This Function

- A result file of magnetic field analysis using an axisymmetric model cannot be referenced.
- The electromagnetic force condition can be set once for one study.
- The electromagnetic force from another frequency with linear interpolation is used for the analysis when the electromagnetic force is not calculated for the frequency specified in [Frequency Control] using the Fast Fourier Transform.

▶▶ Reference

For more information about this condition, see P.801 "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction".

6-13. Temperature Load

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Temperature Load” of the JMAG-Designer User's Manual.

6-13-1. Function

The temperature load condition is used to specify a temperature for parts to run a thermal stress analysis.

A temperature can be set using the results of a thermal analysis (one way coupled analysis).

■ Related items

- Material properties (thermal expansion coefficient, reference temperature)
- Coupled thermal and structural analysis

■ Settings

- Load for part, face, edge, and vertex
- Types of temperatures

■ Available analyses

- Static analysis

■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-13-2. Important Notes Regarding the Use of This Function

- The temperature load condition can be set more than once for one study. However, the temperature load condition cannot be set more than once for one part.
- If the temperature load condition is not set for a part, the thermal expansion of the part is not obtained.
- For a thermal stress analysis, the thermal expansion coefficient and reference temperature need to be set for a material as well as the temperature load condition.

  However, if a thermal load is specified for a part using a temperature difference distribution, the reference temperature specified for the part is ignored.

6-14. Press Fit

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Press Fit” of the JMAG-Designer User's Manual.

6-14-1. Function

The press fit condition is used to obtain a stress distribution caused by a press fit by selecting parts to be press fitted, and then specifying a clearance.

■ Related items

- None

■ Settings

- Clearance(m), (cm), (mm), (μm), (nm), (mil), (ft), (in)

■ Available analyses

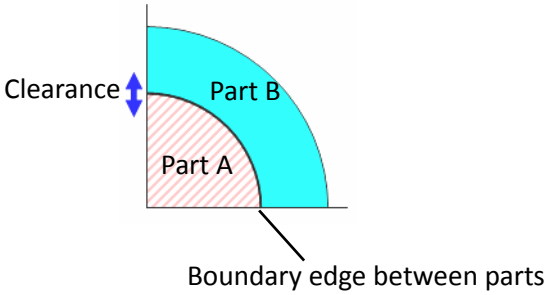
- 2D static analysis

■ Output items

- Surface pressure(Pa)

6-14-2. Important Notes Regarding the Use of This Function

- During a calculation, a beam is created on the edge where the press fit condition is set, and then the pressure on the boundary can be obtained.



Example

6-15. Displacement

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Displacement" of the JMAG-Designer User's Manual.

6-15-1. Function

The displacement condition is used to specify the amount of displacement for the vertices, edges, and faces composing parts.

■ Related items

- Material properties

■ Settings

- Targets
  - When the normal direction is set for a 3D model:
    Faces or shells
  - When the specified direction is set for a 3D model:
    Parts, faces, edges, shells, beams, and vertices.
    Different types of areas, such as faces and edges, can be selected.
  - When the normal direction is set for a 2D model:
    Edges or beams
  - When the specified direction is set for a 2D model:
    Parts, edges, shells, beams, and vertices. Different types of areas, such as faces and edges, can be selected.
- Displacement direction (normal/specified)
- Displacement direction
- Displacement (m), (cm), (mm), (μm), (nm), (mil), (ft), (in)
- Phase (deg) or (rad)

■ Available analyses

- All types of structural analyses

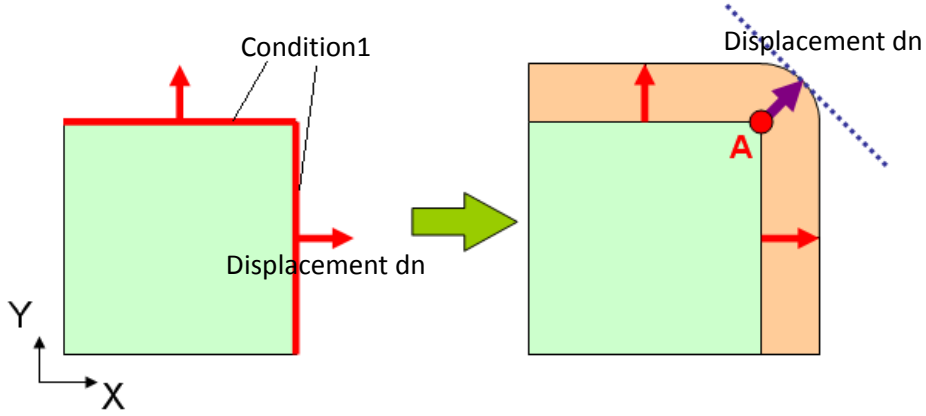
■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

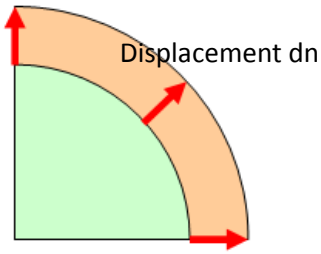
6-15-2. Important Notes Regarding the Use of This Function

- Different types of displacement and phases cannot be set for a vertex with one direction. For example, if displacement condition 1 (displacement direction is X-axis and phase is 5) and displacement condition 2 (displacement direction is X-axis, phase is 10) are set for the same vertex, the analysis cannot be run.
- To constrain the intermediate nodes when the second order elements are used, set the displacement condition for parts, faces, edges, shells, and beams.
- If displacement is specified by the vector, the degree of freedom changes depending on the settings of the condition. For how to set the displacement by the vector, see the following examples.

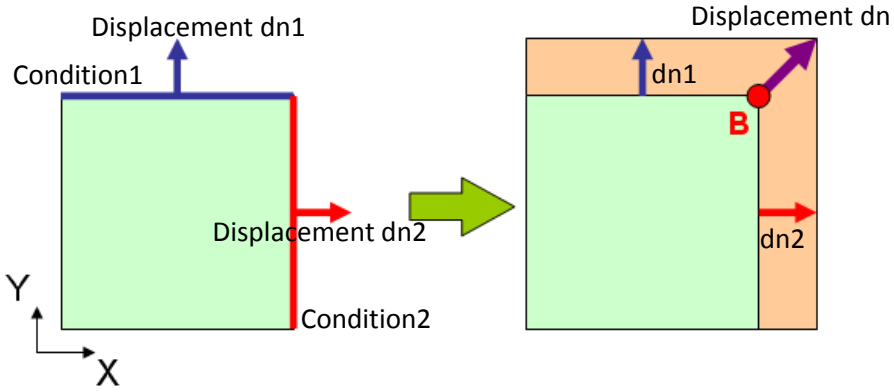
- If two edges are constrained in the normal direction by a displacement condition, the intersection point A has a degree of freedom only in the direction indicated by an arrow, as shown in the following figure. Therefore, point A can move freely within the area bounded by a broken line. The displacement of point A is set to the value specified for the displacement condition.



This example can be applied to the following cylindrical model.



- If two edges are constrained in the normal direction by two displacement conditions, the intersection point B has a degree of freedom only in the Z-axis direction because the X-axis and Y-axis directions are constrained. The displacement at point B is determined by the values specified for two displacement conditions.



6-16. Rigid Body

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Rigid Body” of the JMAG-Designer User's Manual.

6-16-1. Function

The rigid body condition is used to specify areas where the model geometry does not deform. The amount of deformation in the specified areas is not taken into account.

■ Related items

- None

■ Settings

- Constraint for parts, faces, edges, and vertices

■ Available analyses

- All types of structural analyses

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

6-16-2. Important Notes Regarding the Use of This Function

- The different types of areas, such as parts and edges, can be set for one condition.

6-17. Constraint

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Constraint" of the JMAG-Designer User's Manual.

6-17-1. Function

The constraint condition is used to constrain parts, edges, faces, and vertices.

■ Related items

- None

■ Settings

- Constraint
  - When a 3D model is used and all areas are constrained:
    Vertices, edges, faces, and parts
  - When a 3D model is used and the normal or tangent direction is constrained:
    Edges and faces
  - When a 2D model is used and all areas are constrained:
    Vertices, edges, and parts
  - When a 2D model is used and the normal or tangent direction is constrained:
    Edges

■ Available analyses

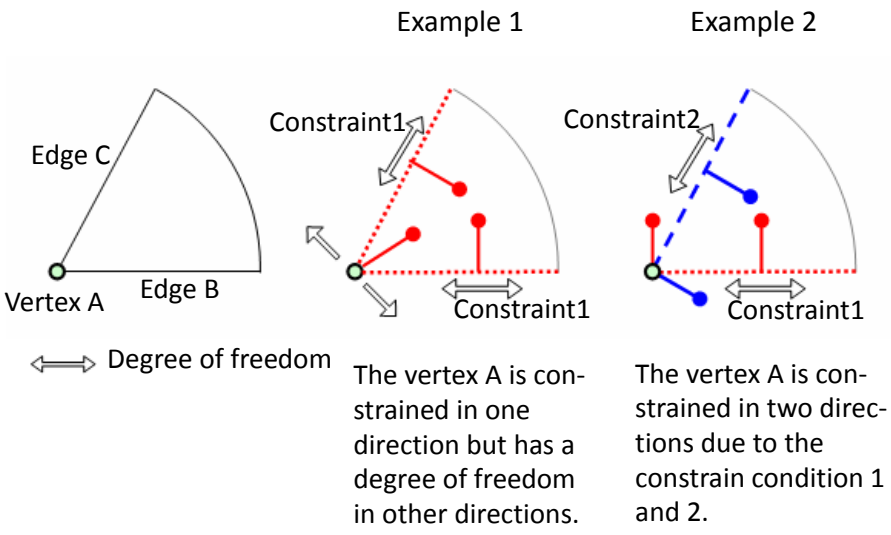
- All types of structural analyses

■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-17-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once for one study.
- Different types of areas, such as faces and edges, can be included in one constraint condition.
- The following must be noted when the constraint condition is set for a 3D model.
  - Only one axis in the normal direction is constrained if the constraint condition is set on the face in the normal direction.
  - Two axes other than in the normal direction are constrained if the constraint condition is set on the face in the tangent direction.
  - Two axes other than in the tangent direction of the edge are constrained if the constraint condition is set on the edge in the normal direction.
  - Only one axis in the tangent direction is constrained if the constraint condition is set on the edge in the tangent direction.
- To constrain the degree of freedom on the periodic boundary in the circumferential direction for a partial model that has a vertex at the origin, the constraint condition needs to be set on two faces or two edges that define the origin.
  For example, the degree of freedom on the periodic boundary is constrained in the circumferential direction as shown in the following figure.
  If the normal direction is constrained by setting the edge B and edge C in one constraint condition as shown in Example 1, the vertex A is constrained in one direction. Therefore, the vertex A has a degree of freedom in other directions.
  If the edge B and edge C are set in different constraint condition as shown in Example 2, the direction to constrain is different between two constraint conditions. As a result, the vertex A is constrained in two directions.


Example 1
Example 2
Edge C
Vertex A
Edge B
Constraint1
Constraint1
Constraint2
Constraint1
Degree of freedom
The vertex A is constrained in one direction but has a degree of freedom in other directions.
The vertex A is constrained in two directions due to the constrain condition 1 and 2.

6-18. Periodic Boundary (Rotation)

▶▶ Reference

See Chapter 32 "Study Properties and Conditions of Structural Analysis" "Rotation Periodic Boundary" of the JMAG-Designer User's Manual.

6-18-1. Function

If a model has a rotational periodicity about an axis, a partial model can be created to reduce the calculation time. The rotation periodic boundary condition needs to be set on the cross section of a partial model. The rotation periodic boundary condition cannot be set more than once.

■ Related items

- None

■ Settings

- 1st periodic boundary face (faces, edges, vertices)
- Rotation periodic boundary
  - A point on the coordinate passing through the rotation axis
  - Direction of the rotation axis
  - Periodic angle

■ Available analyses

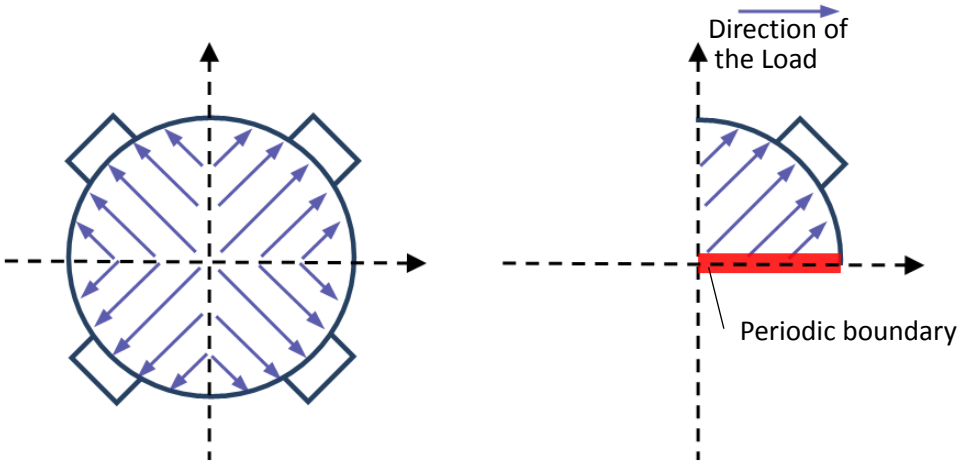
- All types of structural analyses

■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-18-2. Important Notes Regarding the Use of This Function

- The periodic boundary condition is set by taking into account the periodicity of the model geometry and load.
  A full model is shown on the left and a partial model of the full model is shown on the right in the following figure. In this case, the periodic boundary condition is set on the edge of the partial model, as shown in red.



Periodic model (full model) Periodic boundary model (partial model)

- A point on the rotation axis is the center of the coordinate of the rotating model. For example, if a model has a periodicity about the Z-axis and the center of the model is on the origin, the coordinate is (0, 0, 0).
- The direction of the rotation axis needs to be set. For example, if the rotation axis is the Z-axis, the direction of the rotation axis is (0, 0, 1).
- The periodic angle is the center angle of a partial model. For example, if a quarter model is used, the periodic angle is 90 degrees.
- If the rotation periodic boundary is set in a 2D analysis, the rotation axis is the Z-axis.
- When the periodic boundary is set, if a model is rotated for a periodic angle, the amount and direction of physical quantity, such as displacement, is to be the same.
- For a 2D/axisymmetric model, only the edge where a boundary condition is set is regarded as a boundary edge. Therefore, a boundary condition needs to be set to all the edges on the boundary.
  For a 3D/axisymmetric model, only the face where a boundary condition is set is regarded as a boundary face.

6-19. Adhesion

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Adhesion" of the JMAG-Designer User's Manual.

6-19-1. Function

The adhesion condition is used to set an adhesion layer between parts.

■ Related items

- None

■ Settings

- Edge to set the adhesion
- Spring constant (N/m) or (kgf/m)

■ Available analyses

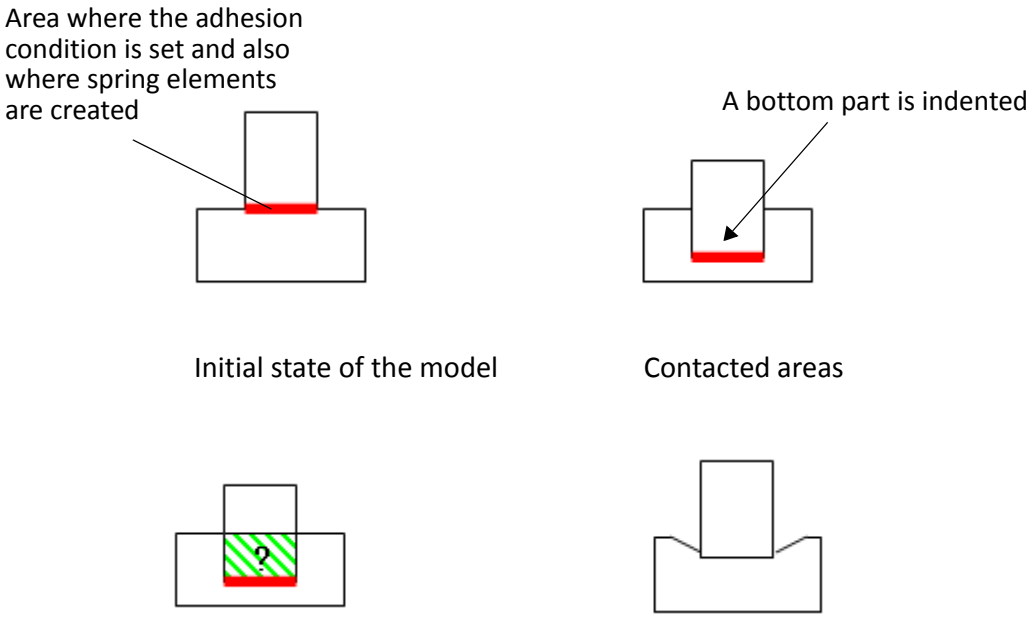
- 2D static analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-19-2. Important Notes Regarding the Use of This Function

- The adhesion condition needs to be set on the contact area of two objects.
- If two objects are separated, the adhesion condition cannot be set.
- The analysis is run twice if areas where the adhesion condition is set have contact. The first analysis evaluates the degree of indentation. The second analysis is run with the shared nodes by removing the spring elements that are indented in the first analysis.


Area where the adhesion condition is set and also where spring elements are created
A bottom part is indented
Initial state of the model
Contacted areas
?

The degree of indentation is evaluated by the first analysis.
The nodes at the indented area are shared in the second analysis.

6-20. Spring

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Spring" of the JMAG-Designer User's Manual.

6-20-1. Function

The spring condition is used to set a spring between two vertices.

■ Related items

- None

■ Settings

- Vertex to set spring
- Spring constant (N/m) or (kgf/m)

■ Available analyses

- All types of structural analysis

■ Output items

- Frequency(Hz)
- Nodal displacement (amplitude, phase)(m, deg)
- Nodal velocity (amplitude, phase)(m/s, deg)
- Nodal acceleration (amplitude, phase)(m/s^2, deg)
- Stress (amplitude, phase)(Pa, deg)

6-20-2. Important Notes Regarding the Use of This Function

- If multiple spring conditions are set for the same pair of vertices, the total spring constant is used for the analysis.

6-21. Concentrated Mass

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Concentrated Mass” of the JMAG-Designer User's Manual.

6-21-1. Function

The concentrated mass condition is used to specify the weight of an object at a vertex. An object can be set as a mass point without modeling the geometry of the object.

■ Related items

- None

■ Settings

- Vertex to define the mass
- Mass (Kg), (g), (mlb)

■ Available modules

- Eigenmode analysis
- Frequency analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (mm, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

6-22. Sound Pressure

▶▶ Reference
See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Sound Pressure" of the JMAG-Designer User's Manual.

6-22-1. Function

The sound pressure condition is used to calculate the sound pressure and sound pressure level at the specified evaluation point.

■ Related items

- Concentrated load
- Pressure load
- Volume load
- Acceleration load
- Centrifugal force
- Electromagnetic force
- Temperature load
- Displacement

■ Targets

- Edges of a 2D model
- Faces of a 3D model

■ Settings

- Evaluation plane of the sound pressure (face or edge)
- Type, size, and position of the evaluation plane
- Coordinate system referenced by the evaluation plane
- Symmetry boundary face

■ Available modules

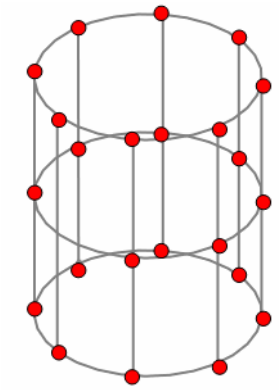
- Frequency analysis

■ Output items

- Sound pressure(Pa)
- Sound pressure level(dB)

6-22-2. Important Notes Regarding the Use of This Function

- To calculate the sound pressure, one or more of the following conditions need to be set.
  - Any of the load conditions listed under [Load] in [Conditions] in JMAG-Designer.
  - A displacement condition in which the displacement is set to greater than 0.
- Shell elements are created on the evaluation plane during a calculation. Nodes are created on the shell elements based on the number of divisions that is specified. Those nodes are used as the evaluation points of sound pressure.



Example) When the evaluate plane is a cylinder, and the number of divisions is set to 8 for [Circumferential] and 2 for [Latitudinal].

6-23. Solver Calculation Control

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Study Properties for Structural Analysis" of the JMAG-Designer User's Manual.

6-23-1. Function

The solver calculation control settings specify the maximum number of iterations for an iterative calculation and the convergence tolerance. Specifying whether or not to use second-order elements or use previous eigenvalue analysis result.

■ Related items

- None

■ Settings

- Flag for using the second element
- Convergence tolerance
- Maximum number of iterations
- Use previous eigenvalue analysis result

■ Available analyses

- All types of structural analysis

■ Output items

- None

6-23-2. Important Notes Regarding the Use of This Function

6-23-2-1. Using Second-order Elements

- When the [Use Second Element] check box is selected, intermediate nodes are created on each element edge and treated as second order elements during the calculation. However, only first order elements are displayed for the results.
- First order elements are not converted to second order elements when the [Use Second Element] check box is selected.
- When the [Use Second Element] check box is selected, the accuracy of calculation is improved but large memory is required. It is not recommended for a large model.
- To constrain the intermediate nodes when the second order elements are used, a displacement condition needs to be set for edges or faces instead of vertices.

6-23-2-2. Referring to Eigenmode Analysis

- This function can be used for the frequency response analysis and the eigenmode analysis.
- This function can refer to eigenmode analysis result files (*.jplot) of the same model when running a frequency response analysis.
  The calculation time can be minimized by referring to an eigenmode analysis result to skip the eigenmode analysis which is included in the process of calculating frequency response analysis.
  This function can recalculate an eigenmode analysis that has been calculated by increasing the number of eigenvalue which then can be added to a result file. The result file that is added can also be referred to a frequency response analysis.

6-24. Eigen Value

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Study Properties for Structural Analysis" of the JMAG-Designer User's Manual.

6-24-1. Function

The eigen value settings specify the number of eigen values (or eigenmodes) to obtain.

In addition, the eigen values (or eigenmodes) of a certain frequency and higher frequencies can be obtained.

In frequency analyses, the number of eigen values (eigenmode) taken into account in the analyses needs to be specified.

■ Related items

- None

■ Settings

- Number of eigen values
- Initial frequency

■ Available analyses

- Eigenmode analysis
- Frequency analysis

■ Output items

- None

6-24-2. Important Notes Regarding the Use of This Function

- Eigen values of the initial frequency and higher frequencies are obtained in ascending order.

6-25. Frequency Control

▶▶ Reference

See Chapter 32. “Study Properties and Conditions of Structural Analysis” “Study Properties for Structural Analysis” of the JMAG-Designer User's Manual.

6-25-1. Function

The frequency control settings specify the frequency used for the analysis.

■ Related items

- None

■ Settings

- Number of steps
- Frequency
- Modal damping ratio

■ Available modules

- Frequency analysis

■ Output items

- None

6-25-2. Important Notes Regarding the Use of This Function

- The number of frequencies is specified for [Number of Steps].
- The frequency analysis is run with the specified frequency. Therefore, frequencies that are specified in the point sequence for the modal damping ratio and concentrated load need to be included in the analysis.

6-25-2-1. Types of Frequency

The following methods can be used to define frequencies.

- Frequency Interval
- Regular Intervals
- Frequency Point Sequence

■ Frequency Interval

The frequency interval specifies the frequency at first step, and the analysis is run at constant interval for the number of steps that is specified.

For example, if the start frequency is 20 Hz, frequency interval is 10 Hz, and the number of steps is 4, the calculation is performed at each step with the following frequencies.

Step 1: 20 Hz

Step 2: 30 Hz

Step 3: 40 Hz

Step 4: 50 Hz

■ Regular Intervals

The regular interval specifies the range of frequencies and divides the range at regular intervals. The analysis is run for the number of steps with the regular intervals from the start frequency in ascending order.

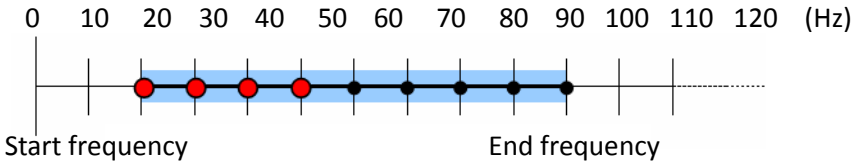
For example, if the start frequency is 20 Hz, end frequency is 100Hz, and the number of divisions is 8, the calculation is performed at each step with the following frequencies.

Step 1: 20 Hz

Step 2: 30 Hz

Step 3: 40 Hz

Step 4: 50 Hz

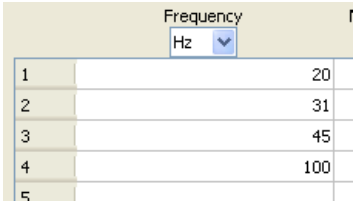


If the number of steps is larger than the number of divisions, the frequency is automatically extrapolated based on the frequency interval calculated from the start frequency, end frequency, and the number of divisions.

For example, in the aforementioned case, the frequency interval is 10 Hz based on the start frequency, end frequency, and the number of divisions. Therefore, if the number of steps is set to 10, the frequency at step 10 is 110Hz.

■ Frequency Point Sequence

The frequency point sequence specifies frequencies at each step. The frequency point sequence is used when the interval of frequency is not constant.



If the number of frequencies specified in the point sequence is larger than the number of steps, the analysis terminates when the number of steps exceeds the number of frequencies.

For example, if the number of frequencies specified in the point sequence is 5 and the number of steps is 6, the calculation after step 5 is not performed.

6-26. Output Control

▶▶ Reference

See Chapter 32. "Study Properties and Conditions of Structural Analysis" "Study Properties for Structural Analysis" of the JMAG-Designer User's Manual.

6-26-1. Function

The output control settings specify the output items to be exported to a result file.

■ Related items

- None

■ Settings

- Output of stress
- Output of strain
- Output of load
- Output of electric potential
- Output of velocity * Only frequency response analysis
- Output of acceleration * Only frequency response analysis

▶▶ Reference

For more information, see P.617 "6-27. Output Items of Structural Analysis".

6-27. Output Items of Structural Analysis

The structural analysis module (DS) can display the following items as analysis results:

MEMO Each of the export items are displayed in units that are applied to an analysis model in JMAG-Designer.

Output item	Unit	Type	Object	Note
Electric potential	V	Scalar	Node	Output when electric potential boundary condition is set.
Displacement	m	Vector	Node	The amount of displacement from the previous step at each node is obtained.
Velocity	m/s	Vector	Node	
Acceleration	m/s^2	Vector	Node	
Global stress	Pa	Tensor	Element	
Principal stress	Pa	Tensor	Element	Tensor history cannot be displayed.
Mises stress	Pa	Scalar	Element	For the calculation formula of mises stress, see MEMO mentioned later.
Stress (cylindrical coordinate system)	Pa	Tensor	Element	The stress is obtained by converting the global coordinate system (X, Y, Z) to the cylindrical coordinate system (r, θ, z).
Sound pressure	Pa	Scalar	Node	Output when sound pressure condition is set.
Sound pressure level	dB	Scalar	Node	The sound pressure level is obtained when the sound pressure condition is set. For the calculation formula of the sound pressure level, see MEMO mentioned later.

Output item	Unit	Type	Object	Note
Global strain	N/A	Tensor	Element	
Principal strain	N/A	Tensor	Element	Tensor history cannot be displayed.
Mises strain	N/A	Scalar	Element	
Face pressure	Pa	Vector	Element, edge	The face pressure is obtained when the press fit analysis is run.
Nodal force (input load)	N * Regard-less of 2D or 3D	Vector	Node	See MEMO mentioned later.

MEMO The calculation formula of mises stress

$$\text{Mises stress } \bar{\sigma} = \sqrt{\frac{1}{2}\{(PS\sigma(I)-PS\sigma(II))^2+(PS\sigma(II)-PS\sigma(III))^2+(PS\sigma(III)-PS\sigma(I))^2\}}$$

PS: Principal stress

MEMO The calculation formula of sound pressure level

$$\text{Sound pressure level SPL} = 20\log_{10}\left(\frac{|P|}{\sqrt{2}P_{ref}}\right)$$

P= Sound pressure at each node (Pa: N/m^2) P_{ref} = 2.0e - 5 (Pa: N/m^2)

MEMO In eigenmode analyses, each item is displayed by taking into account the amplitude and phase.

MEMO In eigenmode analyses, the Y2-axis can be displayed.

MEMO Nodal force

The nodal force is the total sum of loads on each node set by the user. The load acting on each node is calculated based on the following conditions.

- Static analysis: electromagnetic force condition, temperature load, pressure load condition (3D), centrifugal force condition, volume load condition, and acceleration load condition.
- Frequency response analysis: electromagnetic force condition, pressure load condition (3D), and volume load condition.

MEMO Displaying a principle stress tensor or principle strain tensor

- For 2D analyses, the principal stress is displayed by the biaxial compressive and tensile stress vector.
- For 3D analyses, the principal stress is displayed by the triaxial compressive and tensile stress vector.

Chapter 7. Function of Structural Analysis (JMAG-Studio)

This chapter describes in detail the material properties and conditions used in structural analyses.

7-1. Overview of Functions

7-1-1. DS (Structural Analysis Module)

The static, modal, and modal frequency response analyses are available from the JMAG structural analysis module.

■ Static analysis

- The static analysis is used to evaluate displacement and stress when static load acts on a structure.
- A thermal load can be set. The temperature data obtained from a thermal analysis can be used for a structural analysis by using the Temperature for Structural Analysis tool.

■ Modal analysis

- The modal analysis is used to obtain the eigen value and eigen vector of a structure.

■ Modal frequency response analysis

- The modal frequency response analysis is used to obtain the response of a structure that constantly vibrates.
- Displacement, velocity, acceleration, and stress at an arbitrary frequency can be obtained as calculation results.
- Electromagnetic force can be taken into account in a static analysis (DS module) by using the results of a transient response analysis (TR or DP module).
- Electromagnetic force can be taken into account in a modal frequency response analysis (DS module) by using the results of a frequency response analysis (FQ module).
- Electromagnetic force can be taken into account in a modal frequency response analysis (DS module) by using the results of a transient response analysis (TR or DP module) and the fast Fourier transform function.

7-2. Conditions of Structural Analysis (JMAG-Studio)

JMAG-Studio supports the following analysis conditions.

Analysis condition	Module*1		
	DS-ST	DS-EIG	DS-FQ
Executive control section			
Comment ($)	O	O	O
Solver type (SOL)	⊙	⊙	⊙
Parameters for eigen value calculation ($subite, $subtol)*7	X	O	O
End of control section (CEND)	⊙	⊙	⊙
Case control section			
Frequency ID (FREQ)	X	X	⊙
Eigen value ID (METH)	⊙	⊙	⊙
Damping coefficient ID (SDAM)	X	X	O
Dynamic load ID (DLOAD)	X	X	⊙
Single point constraint ID (SPC)	⊙	⊙	⊙
Force ID (LOAD)*2	O	X	X
Heat load ID (TEMPERATURE)*2	O	X	X
Multipoint constraint ID (MPC)	O	O	O
Bulk data section			
Frequency range (FREQ1)	X	X	⊙
Eigen value (EIGRL)	X	⊙	⊙
Damping table (TABDMP1)	X	X	O
Solid element property (PSOLID)*3	O	O	O
Shell element property (PSHELL)*3	O	O	O
Beam element property (PBAR)*3	O	O	O
Isotropic material(MAT1)	⊙	⊙	⊙
Anisotropic material 2D(MAT2, JMAT2)	⊙	⊙	⊙
Anisotropic material 3D(MAT9, JMAG9)	⊙	⊙	⊙
Nodal coordinates (GRID)	⊙	⊙	⊙
6-sided solid element (CHEXA)*4	O	O	O
4-sided solid element (CTETRA)*4	O	O	O
5-sided solid element (CPENTA)*4	O	O	O

Analysis condition	Module*1		
	DS-ST	DS-EIG	DS-FQ
Quadrilateral shell element (CQUAD4)*4	O	O	O
Triangular shell element (CTRIA3)*4	O	O	O
Beam element (CBAR)*4	O	O	O
Dynamic load ID (DLOAD)	X	X	⊙
Dynamic load (RLOAD1)	X	X	⊙
Dynamic load: Scaling factor (DAREA)	X	X	⊙
Dynamic load: Phase angle (DPHASE)	X	X	O
Dynamic load: Table (TABLED1)	X	X	⊙
Force (FORCE)*5	O	X	X
Centrifugal force (RFORCE)*5	O	X	X
Nodal temperature (TEMP)*5	O	X	X
Single point constraint (SPC)	⊙	⊙	⊙
Rectangular coordinate system (CORD2R)	O	O	O
Cylindrical coordinate system (CORD2C)	O	X	O
Cylindrical coordinate system (CORD2C)	O	X	O
Displacement coordinate system (NGRP)	⊙	⊙	⊙
Multipoint constraint (MPC)	O	O	O
Spring element (GSPRING)*6*7	O	O	O
Press fit (PRESS)*7	O	X	X
Concentrated mass (CONM2)	X	O	O
Rigid body element (RBE2)	O	O	O
Adhesion (ASPRING)*7	O	X	X
Acoustic analysis (ACOUST)*7	X	X	O
Electromagnetic force (JLOAD)*7	O	X	O
Piezoelectric constant (MATP)*7	O	O	O
Relative permittivity (MATE)*7	O	O	O
Electrode (ELCRD)*7	O	O	O
Pressure load (PLOAD)	O	X	O
Edge force (LINE FORCE)*7	O	X	O
Volume force (VOLUME FORCE)*7	O	X	O

Analysis condition	Module*1		
	DS-ST	DS-EIG	DS-FQ
Acceleration force (GRAV)	O	X	X
Bulk data end (ENDDATA)	⊙	⊙	⊙

⊙: Required O: Optional X: Not available

*1 Module abbreviations are as follows:

- DS (1): Static analysis
- DS (2): Modal analysis
- DS (3): Modal frequency response analysis

*2 To perform a static analysis, set either of the two conditions.

*3 Set at least one condition

*4 Set at least one condition

*5 To perform a static analysis, set at least one condition.

*6 To perform a press fit analysis, this condition can be set only for a 2D static analysis.

*7 Data cannot be transferred to the NASTRAN data.

Important **DS input files complies with the format defined by MSC/NASTRAN. Information: MSC/NASTRAN, QUICK REFERENCE GUIDE, version 70, Kevin Kilroy, The MacNeal - Schwendler Corporation.**

7-3. Comment $

7-3-1. Function

Comments are specified for a file.

■ Related items

- None

■ Targets

- None

■ Settings

- None

■ Available modules

- All of the modules of structural analysis

■ Output items

- None

7-3-2. Important Notes Regarding the Use of This Function

- The [Comment] box is provided to insert comments in a file.
- Comments can be inserted anywhere in the file.
- All comments are ignored in solver.
- Comments may be up to 80 characters long.

7-4. Solver Type SOL

7-4-1. Function

A solver type to use in a structural analysis is selected.

■ Related items

- None

■ Targets

- Entire model

■ Settings

- Solver type

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-4-2. Important Notes Regarding the Use of This Function

- For a linear static analysis, set sol to 101.
- For a modal analysis, set sol to 103.
- For a modal frequency response analysis, set sol to 111.

7-5. Parameters for Eigen Value Calculation $subite, $subtol

7-5-1. Function

The convergence tolerance and the maximum number of iterations are set for a modal analysis.

■ Related items

- None

■ Targets

- Entire model

■ Settings

- Maximum number of iterations
- Convergence tolerance
- Analysis without constraint

■ Available modules

- DS (modal analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-5-2. Important Notes Regarding the Use of This Function

- An analysis function without any constraints can be used for modal analyses and modal frequency response analyses.

7-6. End of Control Section CEND

7-6-1. Function

The termination of control section is specified.

■ Related items

- Solver type SOL

■ Targets

- None

■ Settings

- None

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-6-2. Important Notes Regarding the Use of This Function

- This condition must always exist in the file.

7-7. Frequency ID FREQ

7-7-1. Function

An ID is assigned to each frequency list condition. The case control condition defines a frequency list condition ID to use in an analysis.

■ Related items

- Frequency range FREQ1

■ Targets

- Frequency ID

■ Settings

- None

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-7-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of FREQ corresponds to that of FREQ1.

7-8. Eigen Value ID METH

7-8-1. Function

An ID is assigned to each eigen value condition. The case control condition defines a eigen value condition ID to use in an analysis.

■ Related items

- Eigen value EIGRL

■ Settings

- Entire model

■ Available modules

- DS (modal analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-8-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of METH corresponds to that of EIGRL.

7-9. Damping Coefficient ID SDAM

7-9-1. Function

The dynamic load condition defines the ID for a dynamic load condition to use in a modal frequency response analysis.

■ Related items

- Modal damping table TABDMP1

■ Targets

- None

■ Settings

- Damping coefficient ID

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-9-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of SDAM corresponds to that of TABDMP1.

7-10. Dynamic Load ID DLOAD

7-10-1. Function

An ID is assigned to each dynamic load condition. The case control condition defines a dynamic load condition ID to use in an analysis.

■ Related items

- Dynamic load RLOAD1

■ Targets

- Entire model

■ Settings

- Dynamic load ID

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-10-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of DLOAD corresponds to that of DLOAD.
- In JMAG-Studio version 9.01901 or earlier, if the dynamic load condition is set on an area where the displacement coordinate system condition is set, the dynamic load condition is taken into account. Therefore, the X, Y, and Z components are replaced with the R, q, and Z components in the case of the cylindrical coordinate system.

  In JMAG-Studio version 9.02001 or later and 9.1, the dynamic load condition is independent of the displacement coordinate system condition, and the X,Y, and Z components are always treated as they are.

7-11. Single Point Constraint ID SPC

7-11-1. Function

An ID is assigned to each single point constraint condition. The case control condition defines a single point constraint condition ID to use in an analysis.

■ Related items

- Single point constraint SPC

■ Targets

- Entire model

■ Settings

- Single point constraint ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-11-2. Important Notes Regarding the Use of This Function

- Make sure that the SPC ID of the case control corresponds to that of the bulk data.

7-12. Force ID LOAD

7-12-1. Function

An ID is assigned to each force condition. The case control condition defines a force condition ID to use in an analysis.

■ Related items

- Force FORCE
- Centrifugal force RFORCE
- Pressure load PLOAD
- Acceleration force GRAV

■ Targets

- Entire model

■ Settings

- Force ID

■ Available modules

- DS (static analysis)

■ Output items

- Nodal displacement (m)
- Stress(Pa)

7-12-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of LOAD corresponds to the following IDs.
  - FORCE ID
  - RFORCE ID
  - PLOAD ID
  - GRAV ID

7-13. Temperature ID TEMPERATURE

7-13-1. Function

SAn ID is assigned to each nodal temperature condition. The case control condition defines a nodal temperature condition ID to use in an analysis.

■ Related items

- Nodal temperature TEMP

■ Targets

- Entire model

■ Settings

- Temperature ID

■ Available modules

- DS (static analysis)

■ Output items

- Nodal displacement (m)
- Stress (Pa)

7-13-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of TEMPERATURE corresponds to that of TEMP.

7-14. Multipoint Constraint ID MPC

7-14-1. Function

An ID is assigned to each multipoint constraint condition. The case control condition defines a multipoint constraint condition ID to use in an analysis.

■ Related items

- Multipoint constraint MPC

■ Targets

- Entire model

■ Settings

- Multipoint constraint ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-14-2. Important Notes Regarding the Use of This Function

- Make sure that the MPC ID of the case control corresponds to the MPC ID of the bulk data.

7-15. Controlling Output Items velo,acce,stre

7-15-1. Function

Output items can be specified with the executive control condition.

■ Related items

- None

■ Targets

- Entire model

■ Settings

- Output items

■ Available modules

- All of the modules of structural analysis

■ Output items

- None

7-15-2. Important Notes Regarding the Use of This Function

- Since the displacement is certain to be exported, no control is available.

7-16. Frequency Range FREQ1

7-16-1. Function

The frequency range to used in an analysis can be specified with the frequency list condition.

■ Related items

- Frequency IDFREQ

■ Targets

- Entire model

■ Settings

- Start frequency (Hz)
- Frequency increments (Hz)
- Number of frequency increments
- Frequency ID

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-17. Eigen Value EIGRL

Reference
For more information, see “Eigen Value” in Chapter 13, “Setting Conditions of Structural Analysis” in the JMAG-Studio User's Manual.

7-17-1. Function

An eigen value to use in an analysis can be specified with the eigen value condition.

■ Related items

- Eigen value ID METH

■ Targets

- Entire model

■ Settings

- Eigen value ID
- Number of eigen values

■ Available modules

- DS (modal analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-17-2. Important Notes Regarding the Use of This Function

- An eigen value closer to 0 is first found.
- If the eigen value condition is not specified in an analysis, the eigen value is automatically set to 1 in the analysis.

7-18. Modal Damping Table TABDMP1

7-18-1. Function

A modal damping table can be defined using the function of frequency.

■ Related items

- Damping coefficient ID SDAM

■ Targets

- Entire model

■ Settings

- Damping coefficient
- Table ID
- Frequency (Hz)

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-18-2. Important Notes Regarding the Use of This Function

- The modal damping is linearly interpolated or extrapolated.
- If only one frequency is specified for the point sequence, the modal damping specified for the frequency is used. The modal damping for other frequencies is set to be zero.

7-19. Solid Element Properties PSOLID

7-19-1. Function

Solid element properties can be defined with the solid element property condition.

■ Related items

- 3D anisotropic material MAT9, JMAT9

■ Settings

- Parts regarded as solid elements
- Solid element property ID
- Physical property type ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-19-2. Important Notes Regarding the Use of This Function

- The solid element property condition must be set in the solver data file.
- The following conditions cannot be set for a rotation component in an area the solid element property condition is set.
  - Multipoint Constraint
  - Rigid Body Element
- The degree of freedom for solid elements is 3 (translational freedom is 3, and rotational freedom is 0).

7-20. Shell Element Properties PSHELL

7-20-1. Function

Shell element properties can be defined with the shell element property condition.

■ Related items

- Anisotropic material 2D MAT2, JMAT2

■ Settings

- Shell element property ID
- Physical property type ID
- Thickness of a board(m)
- Bending stiffness parameter
- Ratio of transverse shear thickness to membrane
- Stress computation (lower plane) (m)
- Stress computation (upper plane) (m)

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, Phase) (Pa, deg)

7-20-2. Important Notes Regarding the Use of This Function

- The shell element property condition must be set in the solver data file.
- The degree of freedom for shell elements is 5 (translational freedom is 3, and rotational freedom is 2 (excluding the rotation about the normal line on the element face)).

7-21. Beam Element Properties PBAR

7-21-1. Function

Beam element properties can be defined with the beam element property condition.

■ Related items

- Isotropic materialMAT1

■ Settings

- Beam element property ID
- Physical property type ID
- Section area (m^2)
- Section secondary moment 1 (m^4)
- Section secondary moment 2 (m^4)
- Torsional constant (m^4)

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-21-2. Important Notes Regarding the Use of This Function

- The degree of freedom for beam elements is 6 (translational freedom is 3, and rotational freedom is 3).

- A beam element is defined as shown in the following figure.

  $\vec{V}$: Vector consisting of point A and the origin

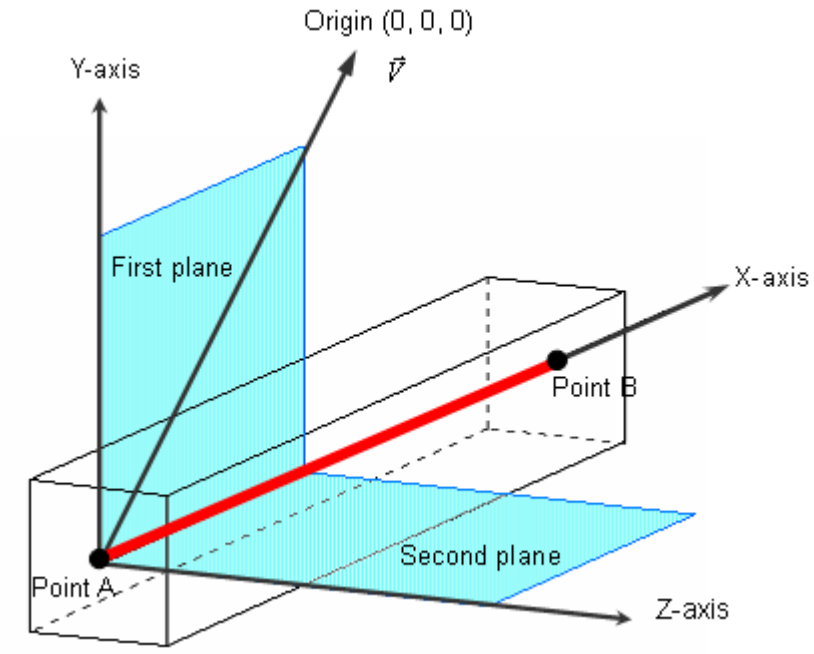
  X-axis: Axis passing through points A and B

  Y-axis: Axis perpendicular to an X-axis, located on the first plane

  Z-axis: Axis perpendicular to an X-axis and Y-axis, passing through point A

  First plane: Plane consisting of an X-axis and $\vec{V}$

  Second plane: Plane parallel to an X-axis and Z-axis



Definition of beam element

The settings for the beam element properties are defined based on the above figure.

- Section secondary moment 1: Moment around a Z-axis
- Section secondary moment 2: Moment around a Y-axis
- Torsional constant: Moment around an X-axis

7-22. Isotropic Material MAT1

7-22-1. Function

Isotropic material properties can be defined with the material settings.

■ Related items

- Solid element property PSOLID
- Shell element property PSHELL
- Nodal temperature TEMP

■ Targets

- Elements

■ Settings

- Physical property type ID
- Young's module (Pa)
- Shear elastic coefficient (Pa)
- Poisson's ratio
- Mass density (kg/m^3)
- Thermal expansion coefficient (1/Celsius)
- Reference temperature (Celsius)

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-22-2. Important Notes Regarding the Use of This Function

- Each term of the following equation needs to be specified.

$$[\sigma] = [K][[\varepsilon] - (T - T_{ref})[A]]$$

  $[\sigma]$: stress, [K]: rigid matrix, $[\varepsilon]$: strain, T: nodal temperature, T_{ref}: reference temperature, [A]: thermal expansion coefficient vector
- The method of defining the material properties is different between 2D analyses and 3D analyses.
- The thermal load is calculated based on the reference temperature specified for the material properties and the nodal temperature specified for the nodal temperature condition.
- The reference temperature here means the temperature around the material. If there is a difference between the reference temperature and the nodal temperature, a material is deformed by the thermal load corresponding to the thermal expansion coefficient. If, for example, the nodal temperature and reference temperature are set to 0 degrees and 20 degrees respectively, the node is under the thermal load of -20 degrees.
- The load is not applied to areas where the nodal temperature condition is not set.

7-23. Anisotropic Material 2D MAT2, JMAT2

7-23-1. Function

2D anisotropic material properties can be defined with the material settings.

■ Related items

- Shell element property PSHELL
- Nodal temperature TEMP

■ Targets

- Elements

■ Settings

- Material ID
- Young's modulus (X-direction) (Pa)
- Young's modulus (Y-direction) (Pa)
- Shear elastic coefficient (XY-direction) (Pa)
- Poisson's ratio (XY-direction)
- Mass density (Kg/m^3)
- Thermal expansion coefficient (X-direction) (1/Celsius)
- Thermal expansion coefficient (Y-direction) (1/Celsius)
- Thermal expansion coefficient (XY-direction) (1/Celsius)
- Reference temperature (Celsius)
- Each component of matrix
- Coordinate system
- Angle from X-axis (deg)

■ Modules in which this function can be used

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)

- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-23-2. Important Notes Regarding the Use of This Function

- Each component of the matrix needs to be specified in the following equation..

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 \\ G12 & G22 & G23 \\ G13 & G23 & G33 \end{bmatrix} \left[\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} A_x \\ A_y \\ A_{xy} \end{Bmatrix} \right]$$

  [σ]: stress, [Gij]: rigid matrix, [ε]: strain, T: nodal temperature, T_{ref}: reference temperature, [A]: thermal expansion coefficient vector, x, y, xy: components of the coordinate system of the material
- The thermal load is calculated based on the reference temperature specified for the material properties and the nodal temperature specified for the nodal temperature condition.
- The reference temperature here means the temperature around the material. If there is a difference between the reference temperature and the nodal temperature, a material is deformed by the thermal load corresponding to the thermal expansion coefficient. If, for example, the nodal temperature and reference temperature are set to 0 degrees and 20 degrees respectively, the node is under the thermal load of -20 degrees.
- The load is not applied to areas where the nodal temperature condition is not set.
- The cylindrical coordinate system cannot be used for anisotropic materials.

7-24. Anisotropic Material 3D MAT9, JMAT9

7-24-1. Function

3D anisotropic material properties can be defined with the material settings.

■ Related items

- Solid element property PSOLID
- Nodal temperature TEMP

■ Targets

- Elements

■ Settings

- Material ID
- Young's modulus (X-direction) (Pa)
- Young's modulus (Y-direction) (Pa)
- Young's modulus (Z-direction) (Pa)
- Shear elastic coefficient (XY-direction) (Pa)
- Shear elastic coefficient (YZ-direction) (Pa)
- Shear elastic coefficient (ZX-direction) (Pa)
- Poisson's ratio (XY-direction)
- Poisson's ratio (XY-direction)
- Poisson's ratio (XY-direction)
- Mass density (Kg/m^3)
- Thermal expansion coefficient (X-direction) (1/Celsius)
- Thermal expansion coefficient (Y-direction) (1/Celsius)
- Thermal expansion coefficient (Z-direction) (1/Celsius)
- Thermal expansion coefficient (XY-direction) (1/Celsius)
- Thermal expansion coefficient (YZ-direction) (1/Celsius)
- Thermal expansion coefficient (ZX-direction) (1/Celsius)
- Reference temperature (Celsius)
- Each component of matrix
- Coordinate system
- Piezoelectric constant (C/m^2)

- Relative permittivity (F/m)

Reference

- For piezoelectric constant, see P.686 "7-52. Piezoelectric Constant MATP".
- For relative permittivity, see P.688 "7-53. Relative Permittivity MATE".

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-24-2. Important Notes Regarding the Use of This Function

- Each component of the matrix needs to be specified in the following equation..

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 & G14 & G15 & G16 \\ & G22 & G23 & G24 & G25 & G26 \\ & & G33 & G34 & G35 & G36 \\ & & & G44 & G45 & G46 \\ & symmetric & & & G55 & G56 \\ & & & & & G66 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \Upsilon_{xy} \\ \Upsilon_{yz} \\ \Upsilon_{zx} \end{Bmatrix} - \begin{Bmatrix} A_x \\ A_y \\ A_z \\ A_{xy} \\ A_{yz} \\ A_{zx} \end{Bmatrix} (T - T_{ref}) \right)$$

  [σ]: stress, [Gij]: rigid matrix, [ε]: strain, T: nodal temperature, T_{ref}: reference temperature, [A]: thermal expansion coefficient vector, x, y, z, xy, yz, zx: components of the coordinate system of the material
- The thermal load is calculated based on the reference temperature specified for the material properties and the nodal temperature specified for the nodal temperature condition.
- The reference temperature here means the temperature around the material. If there is a difference between the reference temperature and the nodal temperature, a material is deformed by the thermal load corresponding to the thermal expansion coefficient. If, for example, the nodal temperature and reference temperature are set to 0 degrees and 20 degrees respectively, the node is under the thermal load of -20 degrees.
- The load is not applied to areas where the nodal temperature condition is not set.
- The cylindrical coordinate system cannot be used for anisotropic materials.

7-25. Nodal Coordinates GRID

7-25-1. Function

The nodal coordinate defines the node ID and coordinates (x, y, z).

■ Related items

- Single point constraint SPC
- Rectangular coordinate systemCORD2R
- Cylindrical coordinate systemCORD2C
- 6-sided solid element CHEXA
- 4-sided solid element CTETRA
- 5-sided solid element CPENTA
- Quadrilateral shell element CQUAD4
- Triangular shell element CTRIA3

■ Targets

- Entire model

■ Settings

- Node ID
- Coordinates for each node (m)

■ Available modules

- All of the modules of structural analysis

■ Output items

- Node ID and coordinates (m)

7-26. 6-sided Solid Element CHEXA

7-26-1. Function

The hexahedron solid element consisting of eight nodes can be used. First order and second order elements can both be created.

■ Related items

- Solid element property PSOLID
- Nodal coordinates GRID

■ Settings

- Element ID
- Solid element property ID
- Connected node ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Nodes

7-26-2. Important Notes Regarding the Use of This Function

- The hexahedron solid elements cannot be used for 2D analyses.
- Even if the [Use Second Order Element] check box is selected in the executive control condition, second order elements are not created. Second order elements are handled internally. Only first order elements are displayed for the results. When the [Use Second Element] check box is selected, the accuracy of calculation is improved but large memory is required. It is not recommended for a large model.

7-27. 4-sided Solid Element CTETRA

7-27-1. Function

The tetrahedron solid element consisting of four nodes can be used. First order and second order elements can both be created.

■ Related items

- Solid element property PSOLID
- Nodal coordinates GRID

■ Targets

- Entire model

■ Settings

- Element ID
- Solid element property ID
- Connected node ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Nodes

7-27-2. Important Notes Regarding the Use of This Function

- The tetrahedron solid elements cannot be used for 2D analyses.
- Even if the [Use Second Order Element] check box is selected in the executive control condition, second order elements are not created. Second order elements are only handled internally. When the results are displayed, they are given as first order elements as well.

7-28. 5-sided Solid Element CPENTA

7-28-1. Function

The pentahedron solid element consisting of six nodes can be used. First order and second order elements can both be created.

■ Related items

- Solid element property PSOLID
- Nodal coordinates GRID

■ Targets

- Entire model

■ Settings

- Element ID
- ID of solid element property
- Connected node ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Nodes

7-28-2. Important Notes Regarding the Use of This Function

- The pentahedron solid elements cannot be used for 2D analyses.
- Even if the [Use Second Order Element] check box is selected in the executive control condition, second order elements are not created. Second order elements are only handled internally. When the results are displayed, they are given as first order elements as well.

7-29. Quadrilateral Shell Element CQUAD4

7-29-1. Function

The isoperimetric membrane-bending quadrilateral shell element consisting of four nodes can be used. First order and second order elements can both be created.

■ Related items

- Shell element properties PSHELL
- Nodal coordinates GRID

■ Targets

- Entire model

■ Settings

- Element ID
- Shell element property ID
- Connected node ID

■ Available modules

- DS (structural analysis module)

■ Output items

- Element ID
- Shell element property ID

7-29-2. Important Notes Regarding the Use of This Function

- The quadrilateral shell elements cannot be used for 3D analyses.
- Even if the [Use Second Order Element] check box is selected in the executive control condition, second order elements are not created. Second order elements are only handled internally. When the results are displayed, they are given as first order elements as well.

7-30. Triangular Shell Element CTRIA3

7-30-1. Function

Defines the isoparametric membrane-bending triangular shell elements. First order and second order elements can both be handled.

■ Related items

- Shell element properties PSHELL
- Nodal coordinates GRID

■ Targets

- Entire model

■ Settings

- Element ID
- Shell element property ID
- Connected node ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Element ID
- Shell element property ID

7-30-2. Important Notes Regarding the Use of This Function

- The triangular shell elements cannot be used for 3D analyses.
- Even if the [Use Second Order Element] check box is selected in the executive control condition, second order elements are not created. Second order elements are only handled internally. When the results are displayed, they are given as first order elements as well.

7-31. Beam Element CBAR

7-31-1. Function

Beam elements can be used.

■ Related items

- Beam element property PBER
- Nodal coordinates GRID

■ Targets

- Entire model

■ Settings

- Element ID
- Beam element property ID
- Connected node ID

■ Available modules

- All of the modules of structural analysis

■ Output items

- Element ID
- Beam element property ID

7-31-2. Important Notes Regarding the Use of This Function

- The beam element properties PBAR must also be defined.
- The nodal temperature condition does not support beam elements.

7-32. Dynamic Load ID DLOAD

7-32-1. Function

An ID is assigned to each dynamic load condition. The case control condition defines a dynamic load condition ID to use in an analysis.

■ Related items

- Dynamic load RLOAD1

■ Targets

- Node

■ Settings

- Dynamic load ID
- Load set ID

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-32-2. Important Notes Regarding the Use of This Function

- Make sure that the ID specified by DLOAD corresponds to the ID of DLOAD.
- Make sure that the ID of RLOAD1 is included in the load set ID specified by DLOAD.

7-33. Dynamic Load RLOAD1

7-33-1. Function

The dynamic load within a frequency range is defined.

■ Related items

- Dynamic load: scaling factor DAREA
- Dynamic load: phase angle DPHASE
- Dynamic load: table TABLED1
- Dynamic load ID DLOAD

■ Targets

- Nodes

■ Settings

- RLOAD1 ID
- DAREA ID
- DPHASE ID
- TABLED1 ID (real part)
- TABLED1 ID (imaginary part)

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-33-2. Important Notes Regarding the Use of This Function

- Make sure that the ID of RLOAD1 is included in the load set ID specified by DLOAD.

7-34. Dynamic Load: Scaling Factor DAREA

7-34-1. Function

The scaling factor of a dynamic load is defined.

■ Related items

- Dynamic load RLOAD1
- Dynamic load: phase angle DPHASE
- Dynamic load: table TABLED1
- Dynamic load ID DLOAD

■ Targets

- Nodes

■ Settings

- DAREA ID
- Node ID
- Load component
- Scaling factor

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-34-2. Important Notes Regarding the Use of This Function

- DAREA should be used along with RLOAD1.

7-35. Dynamic Load: Phase Angle DPHASE

7-35-1. Function

The phase angle is specified for the equation that defines a dynamic load.

■ Related items

- Dynamic load RLOAD1
- Dynamic load: scaling factor DAREA
- Dynamic load: table TABLED1
- Dynamic load ID DLOAD

■ Targets

- Nodes

■ Settings

- DPHASE ID
- Node ID
- Load component
- Phase angle (deg)

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-35-2. Important Notes Regarding the Use of This Function

- DPHASE should be used along with DAREA.

7-36. Dynamic Load: Table TABLED1

7-36-1. Function

The dynamic loads are specified by point sequence.

■ Related items

- Dynamic load RLOAD1
- Dynamic load: scaling factor DAREA
- Dynamic load ID DLOAD

■ Targets

- Nodes

■ Settings

- TABLED1 ID (Real part)
- Frequency (Hz)
- Dynamic load (N)

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-36-2. Important Notes Regarding the Use of This Function

- The frequency and load point sequence data needs to be created.
- The dynamic load is linearly interpolated or extrapolated.
- If only one frequency is specified for the point sequence, the load specified for the frequency is used. The load for other frequencies is set to be zero.

7-37. Force FORCE

7-37-1. Function

Forces (static loads) used for static analysis are defined.

■ Related items

- Force ID LOAD
- Cylindrical coordinate system CORD2C

■ Targets

- Nodes

■ Settings

- Force (first component) (N)
- Force (second component) (N)
- Force (third component) (N)
- Scaling factor
- Cylindrical coordinate system

■ Available modules

- DS (static analysis)

■ Output items

- Nodal displacement (m)
- Stress (Pa)
- Nodal force (N)

7-37-2. Important Notes Regarding the Use of This Function

- Forces (static loads) need to be specified.

7-38. Centrifugal Force RFORCE

7-38-1. Function

A nodal temperature to use in a static analysis can be defined with the nodal temperature condition.

■ Related items

- Force ID LOAD

■ Targets

- Nodes

■ Settings

- Number of rotation (rps)

■ Available modules

- DS (static analysis)

■ Output items

- Nodal displacement (m)
- Stress (Pa)
- Nodal force (N)

7-38-2. Important Notes Regarding the Use of This Function

- A rotation speed needs to be specified.
- The rotation axis is Z-axis.

7-39. Nodal Temperature TEMP

7-39-1. Function

Defines the nodal temperature used for static analysis.

■ Related items

- Thermal load ID TEMPERATURE
- Isotropic material MAT1
- Anisotropic material 2D MAT2, JMAT2
- Anisotropic material 3D MAT9, JMAT9

■ Targets

- Nodes

■ Settings

- Nodal temperature (Celsius)

■ Available modules

- DS (static analysis)

■ Output items

- Nodal displacement (m)
- Stress (Pa)
- Nodal force (N)

7-39-2. Important Notes Regarding the Use of This Function

- An area where the nodal temperature is not set is treated as 0 degrees.
- When setting temperature load, be sure to specify the nodal temperature as well as the reference temperature and thermal expansion coefficient for the material properties.
- The nodal temperature condition cannot be set for beam elements.

7-40. Single Point Constraint SPC

7-40-1. Function

Displacement can be defined by setting the single point constraint condition.

■ Related items

- Single point constraint ID SPC
- 6-sided solid element CHEXA
- 4-sided solid element CTETRA
- 5-sided solid element CPENTA
- Quadrilateral shell element CQUAD4
- Triangular shell element CTRIA3
- Displacement coordinate system NGRP

■ Targets

- Nodes

■ Settings

- Component ID
- Displacement
- Phase
  *The phase setting is available only when the displacement in the normal direction is set in a modal frequency response analysis.

■ Available modules

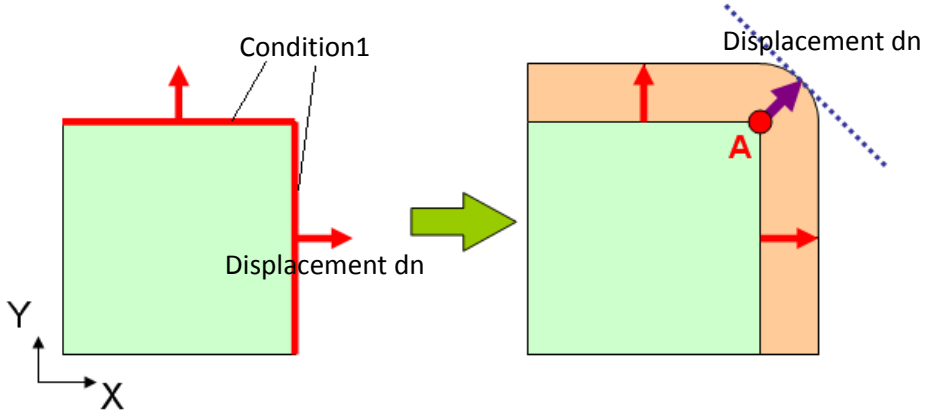
- All of the modules of structural analysis

■ Output items

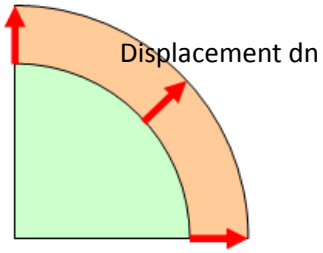
- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-40-2. Important Notes Regarding the Use of This Function

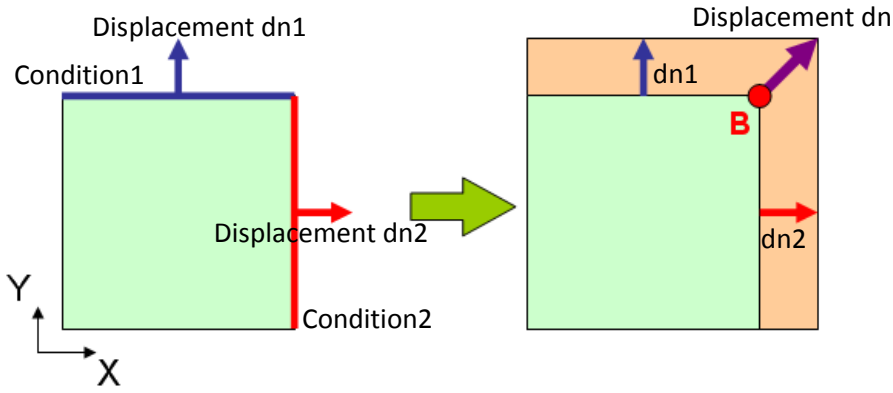
- Make sure that the SPC ID of the case control corresponds to that of the bulk data.
- For example, analysis cannot be run if the single point constraint condition 1 (displacement direction: X-axis, phase: 5) and the single point constraint condition 2 (displacement direction: X-axis, phase: 10) are set for a certain node.
- To constrain the intermediate nodes when the second order elements are used, the single point constraint condition needs to be set not for the node, but for the element edge or the element face.
- If the direction of displacement is specified by vector, the degree of freedom changes according to the setting of the condition. The following is an example for setting the condition.
  - When two edges are constrained in the normal direction by one single point constraint condition, the intersection point A has the degree of freedom only in the direction indicated with an arrow in the figure below. Therefore, point A can move freely within the area bounded by the broken line. The displacement of point A is specified for the single point constraint condition.



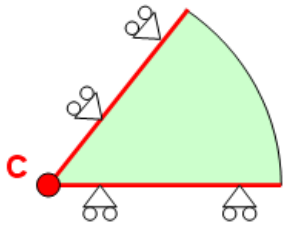
This concept can be applied to a cylindrical model as shown below.



- If two edges are constrained in the normal direction by two single point constraint conditions, the intersection point B has the degree of freedom only in the Z-axis direction because the X- and Y-axis directions are constrained. The displacement at point B is determined by both values specified for the two conditions.



This concept can be applied to the boundary of a partial model as shown below.



Both the X-axis and Y-axis directions are constrained at point C

- If the direction of displacement is specified by the coordinate system, the following should be noted for each element type.
  - For solid elements, the degree of freedom can be set for the translational component in all axis directions. The rotational component is automatically constrained.
  - For beam elements, the degree of freedom can be set for the translational and rotational components in all axis directions.
  - For shell elements, be sure to constrain the rotational component in the normal direction. The degree of freedom can be set for the translational component in all axis directions and the rotational component in two directions.
  - For spring elements, the degree of freedom can be set for the translational component in all axis directions. The rotational component is automatically constrained.

7-41. Rectangular Coordinate System CORD2R

7-41-1. Function

The rectangular coordinate system to use in an analysis can be specified with the coordinate system condition.

■ Related items

- Single point constraint SPC

■ Targets

- None

■ Settings

- Coordinate value

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-42. Cylindrical Coordinate System CORD2C

7-42-1. Function

The cylindrical coordinate system cannot be specified with the coordinate system condition.

■ Related items

- Force FORCE
- Dynamic load RLOAD1
- Displacement coordinate system NGRP

■ Targets

- None

■ Settings

- Coordinate value

■ Available modules

- DS (static analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-42-2. Important Notes Regarding the Use of This Function

- The cylindrical coordinate system cannot be used for anisotropic materials.

7-43. Displacement Coordinate System NGRP

7-43-1. Function

The displacement coordinate system can be defined by setting the displacement coordinate system condition.

■ Related items

- Nodal coordinates GRID
- Rectangular coordinate system CORD2R
- Cylindrical coordinate system CORD2C

■ Targets

- Nodes

■ Settings

- Coordinate value

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-43-2. Important Notes Regarding the Use of This Function

- If the displacement coordinate system condition is set for a node where the following conditions are set, the node is defined by the specified displacement coordinate system. For example, the X and Y components are replaced with R and theta components, respectively if the cylindrical coordinate system is used.
  - Single point constraint condition
  - Multiple constraint condition
  - Rigid body element condition

  The load conditions, such as the electromagnetic force condition, force condition and dynamic load condition, do not use the specified displacement coordinate system.

7-44. Multipoint Constraint MPC

7-44-1. Function

The multipoint constraint equation is defined as follows.

$$\sum_j R_j u_j = 0$$

In this equation, u_j indicates the degree of freedom C_j at the nodal point or scalar point G_j. Furthermore, R_j indicates the scaling factor that the user defines. The degree of freedom (G1,C1) are defined as the dependent degrees of freedom at the beginning of the input column.

■ Related items

- Nodal coordinates GRID

■ Targets

- Node

■ Settings

- Degree of freedom
- Coefficient

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-44-2. Important Notes Regarding the Use of This Function

- When a certain node is specified as a dependent item, the same degree of freedom cannot be set for the node with a multiple constraint condition or rigid body element condition.
- The degree of freedom for the rotation component should not be set to a node of solid element. A rigid matrix cannot be created properly if a node of a solid element is specified.

7-45. Spring Element GSPRING

7-45-1. Function

When the spring element condition is set for nodes comprising a spring element, spring elements are created with the spring constant specified for nodes within the determination distance.

In addition, with this condition, it is possible to select the press fit analysis. In the case of the press fit analysis, spring elements are created between the physical properties set as a target region, and the load corresponding to the amount of dent is defined.

■ Related items

- Nodal coordinates GRID
- Press fit PRESS

■ Targets

- Node

■ Settings

- Spring constant
- Determination distance

■ Available modules

- All of the modules of structural analysis
  * Only a 2D static analysis is use for a press fit analysis.

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-45-2. Important Notes Regarding the Use of This Function

■ When the spring element condition is used:

- Two or more nodes must be selected as a target region.

■ When a press fit analysis is run:

- Nodes set for a target region need to be matched in the circumferential direction.
- The direction of dent is assumed to be the radial direction centered at the origin.

7-46. Press Fit PRESS

7-46-1. Function

The press fit condition is used to create spring elements in the clearance, and define a load corresponding to the clearance.

■ Related items

- Spring element GSPRING

■ Targets

- Element edge

■ Settings

- Clearance (m) *When a clearance is not modeled.

■ Available modules

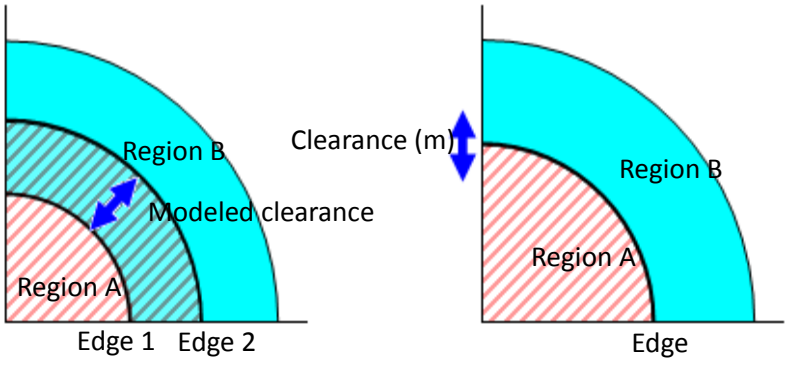
- DS (static analysis) * For a 2D model.

■ Output items

- Pressure (Pa)

7-46-2. Important Notes Regarding the Use of This Function

- A beam element is created on an element edge where the press fit condition is set, and a pressure that acts on the material boundary can be obtained.



- The direction of press fit is defined by the press fitting edge 1.

7-47. Concentrated Mass CONM2

7-47-1. Function

The concentrated mass condition defines a mass for a selected node.

■ Related items

- Nodal coordinates GRID

■ Targets

- Node

■ Settings

- Mass

■ Available modules

- DS (modal analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-48. Rigid Body Element RBE2

7-48-1. Function

The rigid body element condition defines rigid body elements for which the independent degree of freedom is specified for a single node, and the dependent degree of freedom is specified for a certain number of nodes.

■ Related items

- Nodal coordinates GRID

■ Targets

- Node

■ Settings

- Degree of freedom of constraint

■ Available modules

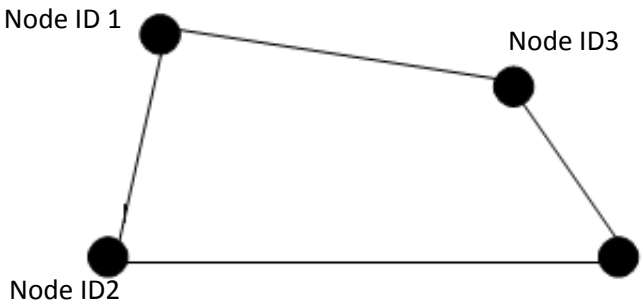
- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-48-2. Important Notes Regarding the Use of This Function

- When a certain node is specified as a dependent node, the same degree of freedom cannot be set for the node with a multiple point constraint condition a rigid body element condition.



Good

Rigid body element condition 1	Rigid body element condition 2
Dependent: X-component of node ID 1 Independent: X-component of node ID 2	Dependent: X-component (Rotation) of node ID 1 Independent: X-component (Rotation) of node ID 3

Wrong

Rigid body element condition 1	Rigid body element condition 2
Dependent: X-component of node ID 1 Independent: X-component of node ID 2	Dependent: X-component of node ID 1 Independent: X-component of node ID 3

In relation to the same node (In this case, node ID 1), the same constraint degree of freedom (In this case, X-component) is set with multiple condition (In this case rigid body element condition 1 and 2)

- The degree of freedom for the rotation component should not be set to a node of solid element. A rigid matrix cannot be created properly if a node of a solid element is specified.

7-49. Adhesion ASPRING

7-49-1. Function

The adhesion condition is used to create spring elements in the region set as a adhesion region.

Furthermore, it is possible to use the simplified contact function. When the simplified contact function is set, an analysis is run two times consecutively. The first analysis is run to obtain the degree of dent for a material on the master side, and the second analysis is run with shared nodes by excluding spring elements in the dented region.

■ Related items

- Nodal coordinates GRID

■ Targets

- Node

■ Settings

- Spring constant
- Simplified contact function
- Material ID on the master side

■ Available modules

- All of the modules of structural analysis
  * The simplified contact function is only supported by static analyses.

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)

7-49-2. Important Notes Regarding the Use of This Function

- The adhesion condition is not available for 3D models.
- The adhesion region needs to be set for a physical property boundary.
- The degree of dent in the R direction is determined for the node on the master side of a spring element that is created inside.
- The simplified contact function is used to simulate the contact between a magnet and a rotor core based on an assumption that a centrifugal force analysis is performed for the rotor of an IPM motor.
- The nodes need to be shared between an adhesion region and adhesion face when the pseudo contact function is used.

7-50. Acoustic Analysis ACOUST

7-50-1. Function

The acoustic analysis condition is used to obtain a sound pressure.

■ Related items

- Electromagnetic force JLOAD

■ Targets

- Node *For 2D analyses
- Element face *For 3D analyses

■ Settings

- Sound pressure evaluating point radius

■ Available modules

- DS (modal frequency response analysis)

■ Output items

- Sound pressure (amplitude, phase) (Pa) *For acoustic analyses
- Sound pressure level (dB) *For acoustic analyses

7-50-2. Important Notes Regarding the Use of This Function

- A JCF file is used to run an acoustic analysis. A solver data file cannot be used for an acoustic analysis.

7-51. Electromagnetic Force JLOAD

▶▶ Reference

- See the JMAG-Studio User's Manual “Chapter 13 Setting Conditions of Structural Analysis”, “Electromagnetic Force”.
- For information about the file to be referenced for magnetic field analysis, see P.801 "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction".

7-51-1. Function

A structural analysis (static analysis / modal frequency response analysis) is run by using electromagnetic force at each node contained in the plot file of a transient response magnetic field analysis. In the modal frequency response analysis, a fast Fourier transform (FFT) is performed.

■ Related items

- Acoustic analysis ACOUST

■ Targets

- Node

■ Settings

- Magnetic field analysis model
  - Input file (Solver data file or JCF file)
  - Plot file
  - Reference interval (starting step, ending step)
  - Full model extension
- Structural analysis model
  - Target
  - FFT resolution
  - Scaling factor
- Flag for using different mesh different a magnetic field analysis and a structure analysis

- Mapping parameter
  - Extension to 2D (magnetic field) to 3D (structure)
  - Extend distance

■ Available modules

- DS (static analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Sound pressure level (dB) *In the case of acoustic analysis
- Nodal force (N)

7-51-2. Important Notes Regarding the Use of This Function

- The electromagnetic force condition needs to be set for a sound analysis.
- From version 8.1, nodal force can be mapped by only setting the electromagnetic force condition.
  Between version 8.1 and 8.3, a control file “fft_cntl” is created in the same directory as the plot file while an analysis is run when the electromagnetic force condition is set.
  From version 8.4, the control file “fft_cntl” is not exported. Contents of the control file “fft_cntl” is included in the JCF file.
- The rotation axis used to create a full model is the Z-axis of the global coordinate system.
- The direction of extrusion from 2D (magnetic field) to 3D (structure) is the Z-axis direction of the global coordinate system.
- Axial symmetric models of magnetic field analyses cannot be used.
- When [Use different mesh between magnetic field and structure] is selected, the electromagnetic force condition needs to be set on nodes of the elements included in at least one layer on the surface. This condition cannot be set only on the nodes on the surface of parts.
- The electromagnetic force from another frequency with linear interpolation is used for the analysis when the electromagnetic force is not calculated for the frequency specified in frequency list conditions using the Fast Fourier Transform.

▶▶ Reference
For more information about this condition, see P.801 "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction".

7-52. Piezoelectric Constant MATP

▶▶ Reference
See the JMAG-Studio User's Manual “Chapter 13 Setting Conditions of Structural Analysis”, “Material Properties”.

7-52-1. Function

The piezoelectric constant settings define piezoelectric constants for a piezoelectric material to perform a piezoelectric analysis.

▶▶ Reference
For piezoelectric analysis, see P.690 "7-54. Electrode ELCRD".

■ Related items

- 3D anisotropic material MAT9, JMAT9
- Relative permittivity MATE
- Electrode ELCRD
- Pressure load PLOAD

■ Targets

- Element

■ Settings

- Piezoelectric constant

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric potential (amplitude, phase) (V, deg)
- Charge density (amplitude, phase) (C/m^3, deg)

7-52-2. Important Notes Regarding the Use of This Function

- Only 3D models are used for piezoelectric analyses.
- A piezoelectric constant is defined by the following equation.

$$\{T\} = [C^E]\{S\} - [e]^T\{E\}$$

[e]: Piezoelectric constant matrix e (C/m^2)

$[C^E]$: Elastic rigidity matrix in constant electric field (Pa)

{T}: Stress vector (Pa)

{S}: Distortion vector

{E}: Electric field intensity vector (N/C)

- For material data, the piezoelectric constant matrix can be expressed by d as follows.

$$[d] = \begin{bmatrix} d_{11} & d_{12} & d_{13} & d_{14} & d_{15} & d_{16} \\ d_{21} & d_{22} & d_{23} & d_{24} & d_{25} & d_{26} \\ d_{31} & d_{32} & d_{33} & d_{34} & d_{35} & d_{36} \end{bmatrix}$$

JMAG can only use the piezoelectric constant matrix [e]. If e is not used, the piezoelectric constant matrix needs to be converted using the following equation.

$$[e] = [d][C^E]$$

$$[e]^T = \begin{bmatrix} e_{11} & e_{12} & e_{13} & e_{14} & e_{15} & e_{16} \\ e_{21} & e_{22} & e_{23} & e_{24} & e_{25} & e_{26} \\ e_{31} & e_{32} & e_{33} & e_{34} & e_{35} & e_{36} \end{bmatrix}^T = \begin{bmatrix} e_{11} & e_{21} & e_{31} \\ e_{12} & e_{22} & e_{32} \\ e_{13} & e_{23} & e_{33} \\ e_{14} & e_{24} & e_{34} \\ e_{15} & e_{25} & e_{35} \\ e_{16} & e_{26} & e_{36} \end{bmatrix}$$

7-53. Relative Permittivity MATE

▶▶ Reference
See the JMAG-Studio Users Manual "Chapter 13 Setting Conditions of Structural Analysis", "Material Properties".

7-53-1. Function

The relative permittivity settings define relative permittivities for a piezoelectric material to perform a piezoelectric analysis.

▶▶ Reference
For piezoelectric analysis, see P.690 "7-54. Electrode ELCRD" in this chapter.

■ Related items

- 3D anisotropic material MAT9, JMAT9
- Piezoelectric constantMATP
- Electrode ELCRD
- Pressure load PLOAD

■ Targets

- Element

■ Settings

- Relative permittivity

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric Potential (amplitude, phase) (V, deg)
- Charge Density (amplitude, phase) (C/m^3, deg)

7-53-2. Important Notes Regarding the Use of This Function

- Only 3D models are used for piezoelectric analyses.
- A permittivity is defined by the following equation.

$$\{D\} = [e]\{S\} - [e^S]\{E\}$$

$$[e^S] = \begin{bmatrix} E_{11} & & \\ & E_{22} & \\ & & E_{33} \end{bmatrix}$$

$\{e^S\}$: Permittivity matrix (F/m)

$[e]$: Piezoelectric constant matrix e (C/m^2)

$\{D\}$: Electrical displacement vector (C/m^2)

$\{S\}$: Distortion vector

$\{E\}$: Electric field intensity vector (N/C)

7-54. Electrode ELCRD

▶▶ Reference
See the JMAG-Studio User's Manual "Chapter 13 Setting Conditions of Structural Analysis", "Electrode".

7-54-1. Function

The electrode condition is used to define electrode and electric potential for a piezoelectric analysis.

A piezoelectric analysis including the effect of a structure and electric field can be performed by specifying piezoelectric constants and permittivities for piezoelectric materials. A variety of sensors using a piezoelectric effect can be analyzed by applying force to piezoelectric materials. A variety of actuators using a inverse piezoelectric effect can also be analyzed by applying a voltage to piezoelectric materials.

■ Related items

- Piezoelectric constant MATP
- Relative permittivity MATE
- Pressure load PLOAD

■ Targets

- Node

■ Settings

- Electric potential (V)
- Amplitude (V)
- Phase (deg)

■ Available modules

- All of the modules of structural analysis

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric potential (amplitude, phase) (V, deg)
- Charge density (amplitude, phase) (C/m^3, deg)

7-54-2. Important Notes Regarding the Use of This Function

- To set an electric potential for a modal frequency response analysis, the amplitude and phase need to be specified.
- Different piezoelectric materials need to be specified for each polar plate.
- Only 3D models are used for piezoelectric analyses.

7-55. Pressure Load PLOAD

▶▶ Reference

- See the JMAG-Studio User's Manual “Chapter 13 Setting Conditions of Structural Analysis”, “Pressure Load”.
- For piezoelectric analysis, see P.690 "7-54. Electrode ELCRD".

7-55-1. Function

The pressure load condition is used to define a pressure load for the face of a solid element.

■ Related items

- Force IDLOAD
- Piezoelectric constant MATP
- Relative permittivityMATE
- Electrode ELCRD

■ Targets

- Element face

■ Settings

- Pressure

■ Available modules

- DS (static analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric potential (amplitude, phase) (V, deg)
- Charge density (amplitude, phase) (C/m^3, deg)
- Nodal force (N) *For 3D analyses only

7-55-2. Important Notes Regarding the Use of This Function

- A pressure load can be specified for only solid elements.
- The direction of pressure is the right-hand screw direction for the incidence of the specified face.

7-56. Edge Force LINE_FORCE

7-56-1. Function

The edge force condition is used to define a pressure applied to a region edge or an element edge.

■ Targets

- Region edge
- Element edge

■ Settings

- Pressure

■ Available modules

- DS (static analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric potential (amplitude, phase) (V, deg)
- Charge density (amplitude, phase) (C/m^3, deg)
- Nodal force (N) * For 3D analyses only

7-57. Volume Force VOLUME_FORCE

7-57-1. Function

The volume force condition is used to define a volume force as a load.

■ Targets

- Element

■ Settings

- Each vector component of volume force (component 1, component 2, component 3)

■ Available modules

- DS (static analysis)
- DS (modal frequency response analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric potential (amplitude, phase) (V, deg)
- Charge density (amplitude, phase) (C/m^3, deg)
- Nodal force (N) * For 3D analyses only

7-57-2. Important Notes Regarding the Use of This Function

- The volume force condition cannot be set for shell elements (triangle/quadrilateral elements).

7-58. Acceleration Force GRAV

7-58-1. Function

The acceleration force condition is used to define an acceleration force as a load.

■ Related items

- Force IDLOAD

■ Targets

- Element

■ Settings

- Each vector component of acceleration (component 1, component 2, component 3)

■ Available modules

- DS (static analysis)

■ Output items

- Frequency (Hz)
- Nodal displacement (amplitude, phase) (m, deg)
- Nodal velocity (amplitude, phase) (m/s, deg)
- Nodal acceleration (amplitude, phase) (m/s^2, deg)
- Stress (amplitude, phase) (Pa, deg)
- Electric potential (amplitude, phase) (V, deg)
- Charge density (amplitude, phase) (C/m^3, deg)
- Nodal force (N) * For 3D analyses only

7-59. End ENDDATA

7-59-1. Function

The end of the input file is specified.

■ Related items

- None

■ Targets

- Entire model

■ Settings

- None

■ Available modules

- All of the modules of structural analysis

■ Output items

- None

7-59-2. Important Notes Regarding the Use of This Function

- This must always be defined at the end of the data file.

7-60. Output Items of Structural Analysis

Structural analysis module (DS) can display the following items as analysis results:

Output item	Unit	Type	Object	Note
Displacement	m	Vector	Node	The amount of displacement from the previous step at each node is obtained.
Velocity	m/s	Vector	Node	
Acceleration	m/s^2	Vector	Node	
Stress	Pa	Tensor	Element	
Principal stress	Pa	Tensor	Element	Tensor history cannot be displayed.
Mises stress	Pa	Scalar	Element	For the calculation formula of mises stress, see MEMO mentioned later.
Stress (cylindrical coordinate system)	Pa	Tensor	Element	The global coordinate system (X, Y, Z) is converted to cylindrical coordinate system (r, θ, z).
Sound pressure	Pa	Scalar	Node	A sound pressure is obtained when acoustic analysis condition is set.
Sound pressure level	dB	Scalar	Node	A sound pressure level is obtained when acoustic analysis condition is set. For the calculation formula of sound pressure level, see MEMO mentioned later.
Pressure	Pa	Vector	Element edge	A pressure is obtained when a press fit analysis is run.
Electric potential	V	Scalar	Node	An electric potential is obtained only when the electrode condition is set.

Output item	Unit	Type	Object	Note
Charge density	C/m^3	Scalar	Node	A charge density is obtained when the electrode condition is set.
Nodal force (input load)	N * Regardless of whether in 2D or 3D	Vector	Node	See MEMO mentioned later.

MEMO The calculation formula of mises stress

$$\text{Mises stress } \bar{\sigma} = \sqrt{\frac{1}{2}\{(PS\sigma(I)-PS\sigma(II))^2+(PS\sigma(II)-PS\sigma(III))^2+(PS\sigma(III)-PS\sigma(I))^2\}}$$

PS: Principal stress

MEMO The calculation formula of sound pressure level:

$$\text{Sound pressure level SPL} = 20\log_{10}\left(\frac{|P|}{\sqrt{2}P_{ref}}\right)$$

P= Sound pressure of each node (Pa; N/m^2), P_{ref} = 2.0e - 5 (Pa; N/m^2)

MEMO In modal frequency response analyses, each item is displayed by taking into account the amplitude and phase.

MEMO In modal frequency response analyses, the Y2-axis can be displayed.

MEMO The Y2-axis display for structural analyses Nodal force

This component expresses the total sum of loads acting on each node set by the user. The load acting on each node is calculated based on the following conditions.

- Static analysis: force condition, electromagnetic force condition, nodal temperature condition, pressure load condition (3D), centrifugal force condition, edge load condition, volume load condition, acceleration load condition.
- Modal frequency response analysis: force condition, electromagnetic force condition, pressure load condition (3D), edge load condition, volume load condition.

Furthermore, this component is exported when the [Load] check box is selected in the executive control condition.

MEMO Subtypes for displaying the principal stress table

When [Results] > [Principal Stress] > [Table] is selected from the menu bar in the result window, four subtypes: [Sigma (I)], [Sigma (II)], [Sigma (III)], and [All] are displayed.

- When [All] is selected, the magnitude of each principal stress s (I), s (II) and s (III), and each component in the X-direction, Y-direction and Z-direction is displayed.
- When σ (I) is selected, only the magnitude of the principal stress is displayed.

MEMO Display of the principal stress tensor

- For 2D analyses, the principal stress tensor is expressed by the biaxial compression and tensile stress vector.
- For 3D analyses, the principal stress tensor is expressed by the triaxial compression and tensile stress vector.

Chapter 8. Function of Electromagnetic Wave Frequency Response Analysis (JMAG-Studio)

This section describes in detail how to set the material properties and analysis conditions used in electromagnetic wave frequency response analyses.

▶▶ Reference

See “Chapter 12. Setting Conditions of Electromagnetic Wave Frequency Response Analysis” of the JMAG-Studio User's Manual.

8-1. Overview of Functions

8-1-1. WV (Electromagnetic Wave Frequency Response Analysis Module)

- This module analyzes the electromagnetic fields of high-frequency waves, where the fields change sinusoidally and the displacement currents become important. Transient response analyses are not supported.
- The finite element method (FEM), in which the electric field is regarded as an unknown variable, is adopted as the analysis method.
- This module can be coupled with the thermal analysis module (HT). The temperature properties of permittivity can also be taken into account in the analysis.

Important **The electromagnetic wave frequency response analysis is only supported by JMAG-Studio.**

▶▶ Reference

For more information about coupled analysis, see P.769 "Chapter 10. Coupled Analysis".

8-2. Conditions of Electromagnetic Wave Frequency Response Analysis

Conditions of the electromagnetic wave frequency response analysis supported by JMAG-Studio are listed in the following table.

Analysis conditions	Module
	WV
Material properties	⊙
Frequency	⊙
Perfect electric boundary (symmetry boundary)	O
Perfect magnetic boundary (natural boundary)	O
Transparent boundary	O
Surface impedance boundary	O
PML boundary (perfect matching layer)	O
Periodic boundary	O
In/out port	O
External field	O
External field (line current)	O
Voltage/current excitation	O
Power calculation	O
Current calculation	O
Voltage calculation	O
Lumped constant component	O
Laser beam	O
Plasma	O
Analysis control	⊙
Control of multigrid method	O
Control of adaptive method	O
Control of output items	O
Far field calculation	O
Equivalent current surface calculation	O

⊙: Required O: Optional X: Not available

8-3. Material Properties

▶▶ Reference

See Chapter 12. “Setting Conditions of Electromagnetic Wave Frequency Response Analysis” of JMAG-Studio User's Manual.

8-3-1. Function

Permeability, permittivity, and electrical conductivity can be set.

■ Related items

- None

■ Settings

- Relative permeability
- Density
- Post output
- Multigrid refinement

■ Output items

- None

8-3-2. Important Notes Regarding the Use of This Function

- The material ID needs to be in consecutive order when there are multiple materials. The analysis cannot be run if the material IDs are not sequential.

8-3-2-1. Electrical Conductivity

- If the electrical conductivity of any material is defined as negative (e.g, -1.000), the elements where the material is assigned are excluded from the analysis.

  This method is useful when the surface impedance method must be used for a model for which the skin depth is extremely thin because of the conductor within the model.

  ▶▶ Reference

  See P.708 "8-7. Surface Impedance Boundary".

8-3-2-2. Post Output

- Only materials for which the [Output] check box of the [Post Output] group box is selected are displayed in the result window.
- Only items related to the material for which the [Output] check box is selected are exported to the plot file.
- The [Output] check box is selected in default.

8-3-2-3. Multigrid Refinement

- The setting becomes effective only when [GMG] (Geometric Multi Grid) is selected in the [Multigrid] tab in the [Analysis Control] dialog box.
- When this item is selected, [Multigrid] under [Matrix Solver] in the [Analysis Control] dialog box is selected.
- When this item is selected and [GMG] is selected under [Type of Multigrid], both of adaptive method and multigrid method are applied.
- When this item is not selected and [GMG] is selected under [Type of Multigrid], the multigrid method is not applied. Only adaptive method is applied.

8-4. Perfect Electric Boundary (Symmetry Boundary)

8-4-1. Function

The perfect electric boundary condition sets a perfect electric conductor wall (symmetry boundary).

The in-plane electric field component on the face where this condition is set becomes zero. In other words, the electric field is perpendicular to the face where this condition is set.

■ Related items

- Transparent boundary

■ Targets

- Element face

■ Settings

- None

■ Output items

- None

8-4-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- Shell elements are generated on the face where the perfect electric boundary condition is set when the analysis is complete. If shell elements are created in advance on the cross section of an analysis model, the cross section can be regarded as a perfect electric boundary.

8-5. Perfect Magnetic Boundary (Natural Boundary)

8-5-1. Function

The perfect magnetic boundary condition sets a perfect magnetic conductor wall (natural boundary).

The in-plane magnetic field component on the face where this condition is set becomes zero. In other words, the magnetic field is perpendicular to the face where this condition is set.

■ Related items

- Perfect electric boundary
- Transparent boundary

■ Targets

- Element face

■ Settings

- None

■ Output items

- None

8-5-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- If no boundary conditions are set on the cross section of an analysis model, the cross section is automatically regarded as a perfect magnetic boundary.

8-6. Transparent Boundary

8-6-1. Function

The transparent boundary condition sets a transparent boundary.

When a plane wave having a specified propagation constant is vertically incident on a surface, the surface transmits the wave without any reflection.

■ Related items

- Perfect electric boundary
- In/out port
- Analysis control

■ Targets

- Element face

■ Settings

- Propagation constant

■ Output items

- None

8-6-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- When the propagation constant is specified to zero (default value), the propagation constant is automatically calculated from the material properties of the face where the transparent boundary condition is set.
- The transparent boundary condition must not be set on faces where the input/out port condition is set.

8-7. Surface Impedance Boundary

8-7-1. Function

The surface impedance boundary condition sets a surface impedance, which has the specified material properties, on a boundary surface.

■ Related items

- Symmetry boundary
- Transparent boundary

■ Targets

- Element face

■ Settings

- Relative permeability
- Electrical conductivity
- Surface impedance

■ Output items

- None

8-7-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- If the skin depth is not thin enough compared to the conductor, the analysis results may not be accurate.
- This condition is not suitable if the equation $\sigma \gg \omega\varepsilon_0$ is not satisfied.
- When setting this condition for a conductor within a model, select either of the following two methods.
  - Delete elements set for a conductor, and set this condition for the boundary surface inside a model.
  - Specify a negative value (e.g., -1.000) for the electrical conductivity in the material properties of a conductor, and set this condition for the boundary surface of a conductor or an area in contact with the conductor.

8-8. PML Boundary (Perfect Matching Layer Boundary)

8-8-1. Function

The PML boundary condition sets a perfect matching layer boundary.

A wave absorbing layer absorbs more waves than the transparent boundary, especially, for waves entering with an angle.

■ Related items

- Transparent boundary condition

■ Targets

- Elements

■ Settings

- Absorption direction
- Parameters for absorption (usually use default values)
- Output flag

■ Output items

- None

8-8-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- This condition is set for elements.
- When this condition is set, materials are set for each setting group of the PML conditions, and the results are exported as a block for each setting group.
- The post output can be set with the material properties; however, the post output setting for the PML conditions has priority over the material property setting.

8-8-2-1. Absorption Direction

- When the absorption direction is specified, the higher orders of scattered waves in the specified direction are also absorbed. It is recommended that two or more layers be specified as absorption layers.

8-8-2-2. Absorption Parameter

- The default values can be used for the absorption parameters for normal calculations.

8-9. Periodic Boundary

8-9-1. Function

The periodic boundary condition sets a periodic boundary to reduce the size of an analysis model when an analysis target has a rotational periodicity about an axis or a translational periodicity in the vector direction.

■ Related items

- None

■ Targets

- Nodes on the first periodic boundary

■ Settings

- Periodic /antiperiodic boundary
- Direction of translational periodicity
- Periodic distance

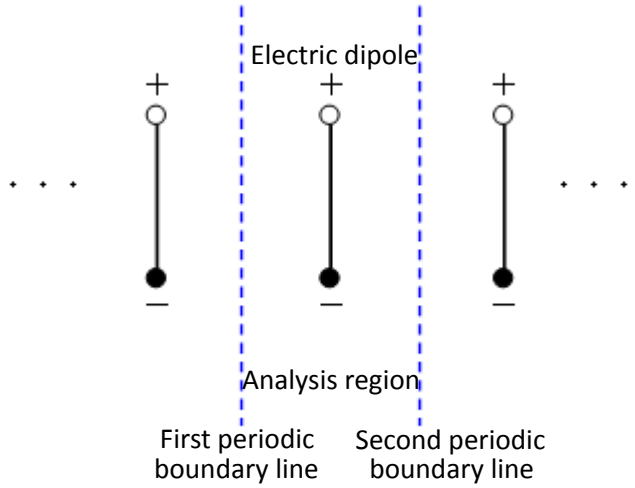
■ Output items

- None

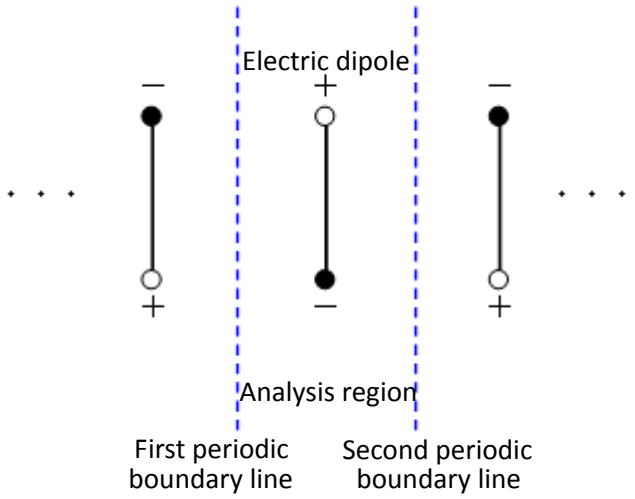
8-9-2. Important Notes Regarding the Use of This Function

- The rotational periodicity settings are not currently supported.

- Select the nodes on the first periodic boundary surface in the analysis model.



Periodic boundary



Antiperiodic boundary

- The program automatically locates the nodes on the second periodic boundary surface.
- Nodes on the second periodic boundary surface must have corresponding nodes on the first periodic boundary surface.

8-9-2-1. Periodic Boundary/Anti-periodic Boundary

- For a periodic boundary, when a model is moved for a periodic distance, the physical volume such as magnetic flux density remains the same in amplitude and direction.
- For a antiperiodic boundary, when a model is moved for a periodic distance, the physical volume such as magnetic flux density remains the same in amplitude, but the direction is reversed.

8-9-2-2. Translational Periodic Boundary

- "Translation direction" is the vector representing the direction of translation. For example, if the boundary has a translational periodicity in the X-axis direction, the direction is (1, 0, 0).
- For the translational periodic boundary, enter the periodic distance instead of the periodic angle. For example, if it has a periodicity every 30 cm, the periodic distance is 0.3 m.

8-10. In/Out Port

8-10-1. Function

The in/out condition sets the mode (propagation constant) and intensity of incident waves for a specified port.

Incident waves are defined by plane wave, user subroutine, modal port (rectangular wave guide, cylindrical wave guide, cylindrical coaxial cable, and microstrip line) or plane wave (oblique plane wave).

■ Related items

- Transparent boundary boundary
- User subroutine: *incusr*

■ Targets

- Element faces

■ Settings

- Specification method
- Modal identification number
- Power amplitude
- Propagation coefficient
- Excitation flag
- User subroutine: *incusr*

■ Output items

- None

8-10-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.

8-10-2-1. Excitation Mode

- With the excitation flag turned off, it is treated as the output port. Even if an incident wave is specified, incidence is not performed.
- In [Excitation Mode] box, specify the incident mode. The modes are numbered, e.g., 0, 1, 2,… in ascending order of eigen frequency.

  For example, if TE10 is set for the excitation, set 0 for the excitation mode.

8-10-2-2. Natural Modes

- The order of natural modes depends on the port geometry.

  For example, when the aspect ratio b/a is 1/2< b/a<1, TE modes are of the order of "TE10, TE01, TE11, TE20." However, if it is 1/3<b/a<1/2, then TE modes are "TE10, TE20, TE01, TE11."

8-10-2-3. User Subroutine

- When using a user subroutine, the function name is *incusr* + (absolute value of modal identification number).

  For example, when setting the modal identification number to –10, the function with the name *incusr* 10 is called.
- User subroutines are written in the Python script language. The script needs not to be recompiled (or reconfigure the module) after changes are made to the script.

  ▶▶ Reference

  For details on how to write the user subroutine, see the sample file or the help file included with Python.

8-10-2-4. Power Amplitude

- The value of the electric power at a specified port includes the value of scattered waves after the calculation is complete. It is generally different from the value specified prior to the calculation.
- If the electric power is not specified, the calculation is performed as the incident power is approximately 1W.
- If the electric power is not specified but a user subroutine is used, the electric power is calculated from the value of electric field specified in the subroutine.
- If a user subroutine is used and the electric power is specified, the field is calculated from the specified electric power.

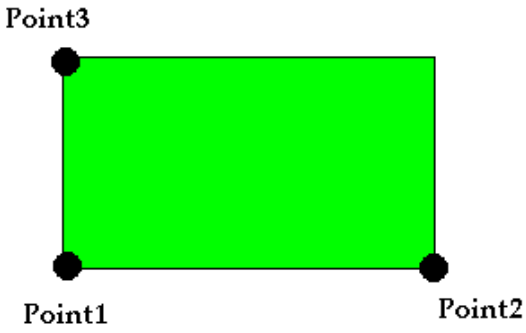
8-10-2-5. Modal Port

- When using a modal port (rectangular wave guide, cylindrical wave guide, cylindrical coaxial cable, or microstrip), dimension and mode, and dielectric layer (for microstrip) must be specified.
- Note the following when selecting [Rectangular Wave Guide] or [Cylindrical Wave Guide].
  - Specify the order of mode that passes through the specified in/out port in the [TE] and [TM] boxes. The modes with the order less than or equal to the specified order can propagate the cross section.
    For example, if the aspect ratio b/a of a port cross section is 1/2<b/a<1 and [TE] is set to 3, the modes TE10, TE01, and TE11 can propagate the cross section in ascending order of eigenfrequency.
- Note the following if selecting [Coaxial Cable].
  - Do not model inner-conductor parts. The model needs to be made up only of a dielectric.
  - Specify an electric wall to the face of an object in contact with the inner-conductor and outer-conductor.
  - Specify a point on the inner diameter and a point on the outer diameter from the axial center so that the direction vectors of the two points are not parallel to each other. This is because the direction vectors of the inner and outer radii from the axial center are used as the direction of a wave number vector to express the direction of the outer product.
- Note the following if selecting Microstrip Line.
  - Specify the same material properties for all base dielectrics.
  - Model the strip line with shell elements.
  - Only one dielectric layer can be specified. The multilayer structure cannot be used for a dielectric.

8-10-2-6. Dimension of Modal Port

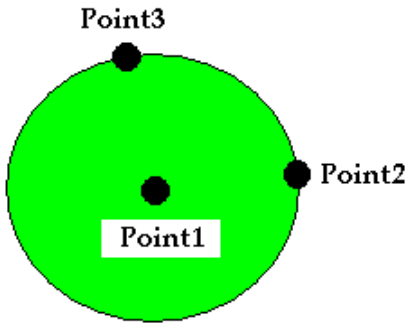
■ Rectangular Wave Guide

- point 1: Origin of the rectangular wave guide
- point 2: A point on the rectangular corner (width direction)
- point 3: Another point on the rectangular corner (height direction)



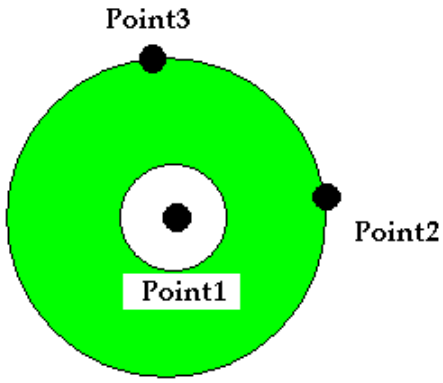
■ Cylindrical Wave Guide

- point 1: Origin of the circular wave guide
- point 2: A point on the circumference of the cylindrical wave guide
- point 3: Another point on the circumference of the cylindrical wave guide



■ Cylindrical Coaxial Cable

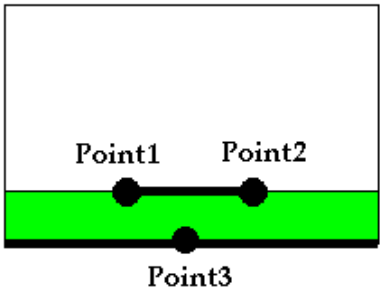
- point 1: Origin of the coaxial cable (center)
- point 2: A point on the inner circumference of the coaxial cable
- point 3: Another point on the outer circumference of the coaxial cable



Important **Make sure that point 1, point 2, and point 3 are not aligned in a straight line.**

■ Microstrip Line

- point 1: A point on the microstrip line
- point 2: Another point on the microstrip line
- point 3: A point on the ground plane



8-11. External Field

8-11-1. Function

The external field condition sets the electric field applied to an entire model, and calculates only scattered waves scattered by dielectrics and conductors. The electric field applied to an entire model is referred to as the "external field."

Total field including the external field and scattered waves or only the scattered waves can be selected to export to the analysis results.

■ Related items

- External field (line current)
- User subroutine: *extusr*

■ Targets

- Entire model

■ Settings

- Direction of external field (plane wave)
- Direction and altitude (vector) of polarized wave of external field (plane wave)
- Usage flag for user subroutine *extusr*

■ Output items

- None

8-11-2. Important Notes Regarding the Use of This Function

- To specify an external field, either the plane wave or the user subroutine can be selected.
- When using the user subroutine, the function is called with the name *extusr* (745 page). As in the case with the in/out port condition, the user subroutine is written in the Python script language.
- This condition cannot be used together with the external field (line current), voltage/current excitation, and in/out port conditions.
- The multigrid method is not available.

8-12. External Field (Line Current)

8-12-1. Function

The external field (line current) condition is used to calculate the electromagnetic wave generated by the line current and apply it to an finite element method model. The beam element of the line current can be defined independently of the finite element model.

■ Related items

- External field condition

■ Targets

- Beam elements

■ Settings

- Current

■ Output items

- None

8-12-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- The entire line current part must be defined regardless of the symmetry of a finite element method model.

8-13. Voltage/Current Excitation

8-13-1. Function

The voltage/current excitation condition sets electric fields or currents for the specified beam element.

■ Related items

- None

■ Targets

- Beam element

■ Settings

- Electric field/current

■ Output items

- None

8-13-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- This condition can be set for the beam elements on the element edges of an analysis model.
- The beam elements that are set by this condition must be aligned in a straight line.

8-14. Power Calculation

8-14-1. Function

The power calculation condition is used to calculate the power transmitted through a specified face.

■ Related items

- In/out port condition

■ Targets

- Element face

■ Settings

- None

■ Output items

- Transmitted power

8-14-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- This condition must be specified for one of two element faces that are adjacent each other. When both element faces are selected, the transmitted power cannot be calculated correctly.

8-15. Current Calculation

8-15-1. Function

The current calculation condition is used to calculate the current transmitted through the face by integrating the magnetic field on the specified beam elements. The element face is specified by enclosing it with beam elements or selecting the element face directly.

■ Related items

- In/out port condition

■ Targets

- Element

■ Settings

- None

■ Output items

- Current

8-15-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.

8-16. Voltage Calculation

8-16-1. Function

The voltage calculation condition is used to calculate the voltage by integrating the electric field on the specified beam elements.

■ Related items

- In/out port condition

■ Targets

- Element

■ Settings

- None

■ Output items

- Voltage

8-16-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- With this condition, a file with the same name as the plot file and an extension of.voltage is exported.
  - The output items are (from the left): 1) frequency (Hz), 2) real voltage (V), and 3) imaginary voltage (V).
  - If multiple voltage calculation conditions are specified, as many values of the real and imaginary voltage as the number of conditions are exported on the right.
  - The output data is delimited by commas every 20 columns.

Frequency	Imaginary voltage of voltage calculation condition 1	Real voltage of voltage calculation condition 1	Real voltage of voltage calculation condition 2	Imaginary voltage of voltage calculation condition 2
4.7400000632E+014,	−1.6471542859E+000,	−3.4397786889E+000,	1.0163587369E+000,	1.5384173836E+000

Example

8-17. Lumped Constant Component

8-17-1. Function

The lumped constant component condition is used to set the impedance, resistor, coil, or capacitor for beam elements.

■ Related items

- None

■ Targets

- Element

■ Settings

- Impedance (real part, imaginary part)
- Resistor
- Coil
- Capacitor

■ Output items

- None

8-17-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- When one component is specified across multiple beam elements, the value of a component is divided by each element. For example, if a resistance of 50 ohm is specified for 5 beam elements that have the same length, each beam element has a resistance of 10 ohm. This value changes depending on the length of elements. If a resistance of 50 ohm is specified for two beam elements with a length ratio of 1:4, one has a resistance of 10 ohm and the other has a resistance of 40 ohm.
- For capacitors, if a capacitor is specified across multiple elements, the capacitance is divided by each element so that the value is the same as when capacitors are connected in series.

8-18. Laser Beam

▶▶ Reference

See Chapter 12. "Setting Conditions of Electromagnetic Wave Frequency Response Analysis", "Laser Beam" of the JMAG-Studio User's Manual.

8-18-1. Function

The laser beam condition sets a laser beam with the Gaussian distribution as an external field. This condition is used to calculate scattered fields generated from the interactions between material objects including the external field.

The laser beam can be specified by the beam of vector fields, beam of scalar fields, or external file.

■ Related items

- External field condition

■ Targets

- Entire model

■ Settings

- Laser beam type
- Propagation vector
- Polarized wave vector
- Focal position
- Spot size
- Numerical aperture
- Output
- Electric field intensity
- Aberration intensity
- Aberration angle
- File name of external field (in case of reading file)

■ Output items

- Electric field intensity(V/m)
- Electric force density (W/m^3)

8-18-2. Important Notes Regarding the Use of This Function

- This condition cannot be used together with the external field condition.
- Multiple laser light sources cannot be set.
- The vector field can be set when three polarized wave components of the electric field exist perpendicular to the moving direction.
- The scalar field can be set when only one polarized wave component of the electric field exist perpendicular to the moving direction.
- The external file can be set by converting the analysis results obtained from the optical simulation software DIFFRACT to the defined format.

8-19. Plasma

▶▶ Reference
See Chapter 12. "Setting Conditions of Electromagnetic Wave Frequency Response Analysis", "Plasma" of the JMAG-Studio User's Manual.

8-19-1. Function

The plasma condition is used to calculate the plasma density by using the electric field value.

■ Related items

- User subroutine: *plsusr_ini, plsusr, epsusr*

■ Targets

- Element

■ Settings

- Plasma calculation region
- Plasma generating region
- Ungenerated region of plasma
- Maximum iteration of plasma calculation
- Convergence tolerance of plasma calculation
- Flag to use a plot file for initial plasma
- Various parameters (within the user subroutine)

■ Output items

- Plasma density($1/m^3$)
- Quality factor

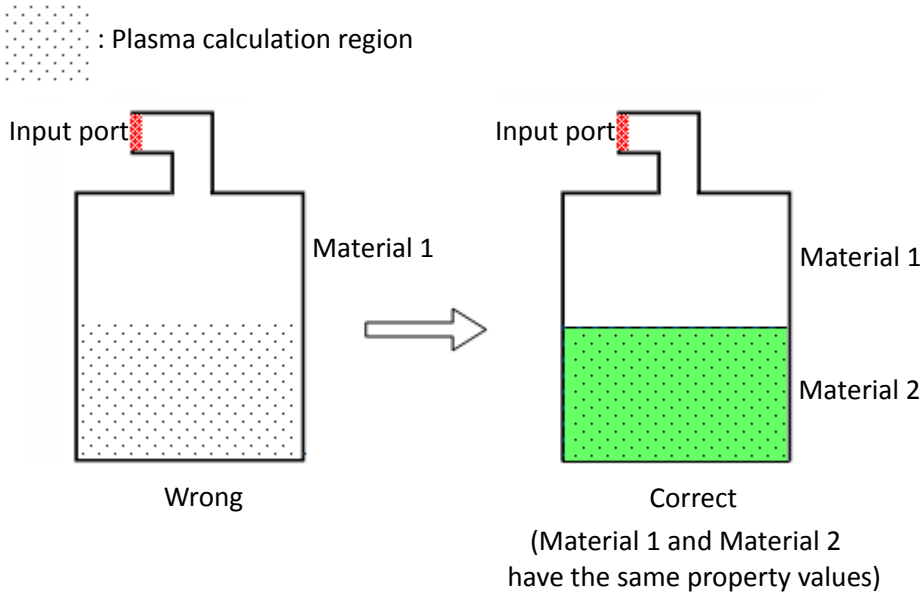
8-19-2. Important Notes Regarding the Use of This Function

- This condition is used together with user subroutines (*plsusr_ini, plsusr, epsusr*). A plasma model can be created by changing the descriptions of these subroutines.

  ▶▶ Reference

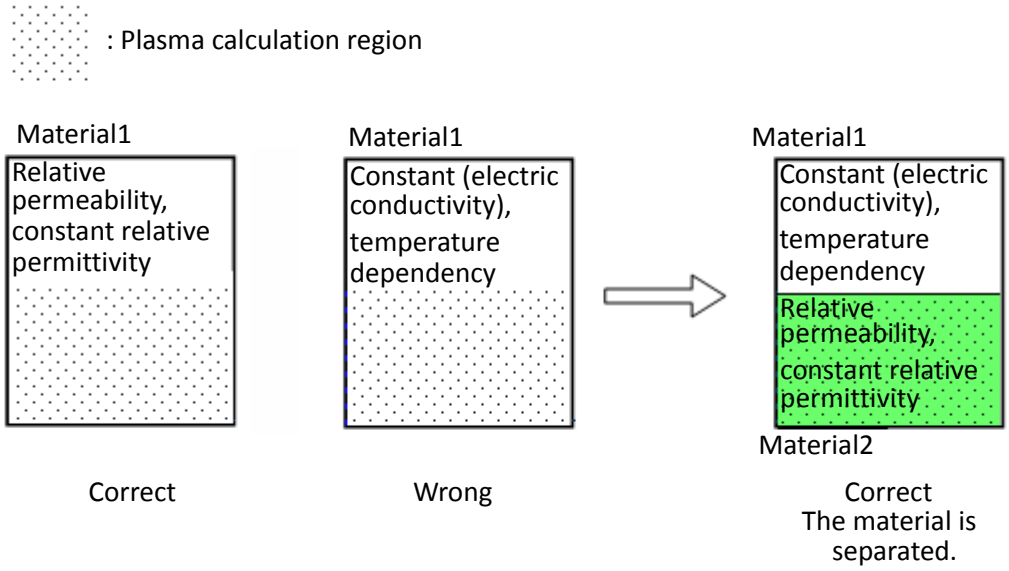
  See P.747 "8-23-7. plsusr_ini", P.749 "8-23-8. plsusr", P.753 "8-23-9. epsusr".

- A plasma calculation region needs to be assigner to the elements to perform a plasma calculation. However, the input/output ports cannot be specified inside the plasma calculation regions. In such a case, multiple materials having the same properties can be created and assigned to the area where the in/out port condition is set and the plasma calculation regions.



Plasma calculation region and material specification (1)

- Only materials having constant relative permeability and relative permittivity can be set in the plasma calculation regions.
  Materials are assigned as follows.



Plasma calculation region and material specification (2)

- When the plasma condition is set, multiple field calculations are performed. The calculation can be controlled by setting the maximum number of iterations and convergence tolerance (the rate of change in total plasma density). The result of the final step after convergence as the plasma density.

- The quality factor Q_f is given as follows:

$$Q_f = \int_{(PlasmaGeneratingRegion)} E^2 dv / \int_{(UngeneratedRegionofPlasma)} E^2 dv$$

where

E : Electric field $\int dv$: Volume integral

8-20. Frequency (Step)

8-20-1. Function

The frequency condition specifies frequencies to use in an analysis. There are three ways to set frequencies.

- Point sequence
  Defines the frequency point sequence. Calculations are performed at the specified frequencies.
- Regular-interval sweep
  Defines the frequency region and the number of divisions. Calculations are performed at the calculation points that are extracted by dividing the specified frequency region by the specified number of divisions.
- Automatic sweep
  Defines the frequency region. Calculations are performed at the calculation points that are automatically extracted from the specified frequency region.

■ Related items

- None

■ Targets

- Analysis model

■ Settings

- Frequency
- Number of divisions
- Start frequency of sweeping
- End frequency of sweeping
- Maximum number of expansion
- Maximum term
- Save expansion term

■ Output items

- None

8-20-2. Important Notes Regarding the Use of This Function

- This condition can be set more than once.
- When the sweep option is used, the two points, the first frequency and the final frequency need to be set.

  When the regular-interval sweep option is used, calculation points are extracted by dividing the frequency region between the first frequency and the final frequency by the specified [Number of Divisions]. An analysis is performed at each calculation point.

  When the automatic sweep option is used, calculation points for [Maximum Number of Expansion] are automatically extracted from the frequency region between the first frequency and the final frequency. An analysis is performed at each calculation point.
- The maximum number of terms means the order of expansion to interpolate the frequency interval using the Pade approximation.

8-21. Analysis Control

Reference
See “Chapter 12. Setting Conditions of Electromagnetic Wave Frequency Response Analysis” “Analysis Control” of the JMAG-Studio User's Manual.

8-21-1. Function

- The analysis control condition sets flags and parameters for an analysis.

■ Related items

- None

■ Settings

- Types of matrix solver
  - Direct method
  - Iterative method
  - Multigrid method
- Settings of direct method
  - In core
  - Out of core
- Types of iterative method
  - Conjugate gradient method (CG method)
  - GCR method
  - GMRES method
- Types of preprocessing
  - Not used
  - SSOR method
- Parameters for iterative matrix solver
  - Acceleration coefficient
  - Convergence tolerance
  - Number of iterations
  - Restart parameter
- Parameters for multigrid method
  - Multigrid type
  - Specifying usage level of direct method

  - Smoothing steps
  - Cycle type
  - Maximum number of nodes for coarse mesh.
- Parameters for adaptive method
  - Adaptive maximum level of divisions
  - Specifying termination error

8-21-2. Important Notes Regarding the Use of This Function

8-21-2-1. Types of Matrix Solver

The following methods are available for the matrix solver.

■ Direct method

The direct method performs a calculation and deletes matrix elements at the same time.

Since all matrix elements are stored, a large amount of memory is required. However, the processing procedures are predetermined. A definitive solution can be obtained in a given time without worrying about the convergence.

■ In core

The entire matrix is stored and processed in the internal memory.

■ Out of core

An external memory is used simultaneously. Routines not having to do directly with a calculation are temporarily saved in an external memory. This method is applicable to a large model.

■ Iterative method

The iterative method processes the matrix for each iteration in accord with the iteration procedures. The iterative method can deal with a larger calculation than the direct method because less memory is required in comparison with the direct method.Note that there may be difficulty with convergence depending on a model.

■ Multigrid method

Multigrid method supports three types: GMC method (geometric multigrid method), AMG method (algebraic method), and Hybrid method (GMC method plus AMG method).

■ GMG method

The GMC method provides the same function as the multigrid method that is supported by Version 6.0 or later versions. Combined with the adaptive method, this method starts with coarse mesh. A large-scale calculation can be performed in high speed with high accuracy. This method cannot presently be used for calculating scattering fields (external fields and external fields generated by line current). This method presently supports only tetrahedron elements. Default parameters can normally be used.

■ AMG method

Contrary to the GMG method, the AMG method starts with dense mesh.A high speed calculation can be performed because the solver processes coarse mesh at the matrix level and speeds up the convergence of an error.

■ Hybrid method

The hybrid method is a combination of the GMG and AMG methods. At present, this method is not supported.

8-21-2-2. Types of Iterative Method

The following methods are available for the matrix iterative solver.

■ CG method

The CG method is used for a normal iterative calculation. The convergence may be slightly poor when the multigrid method is used for the matrix solver.

■ GCR method

The GCR method has superior convergence to the CG method when the multigrid method is used for the matrix solver. However, this method requires a larger amount of memory than the CGT method.

■ GMRES method

The GMRES method has superior convergence to the CG method when the multigrid method is used for the matrix solver. However, this method requires a larger amount of memory than the CGT method.

8-21-2-3. Preprocessing Types

The following options are available for the preprocessing method.

■ Not used

The preprocessing is not necessary when the direct method is used.

■ SSOR method

The SSOR method is used when the normal iterative method is used. The iterative method can be used without preprocessing, but it affects the convergence.

8-21-2-4. Parameters for Iterative Matrix Solver

The following parameters can be set for the iterative method of simultaneous linear equations.

■ Acceleration Coefficient

The acceleration coefficient needs to be changed if the iterations do not converge. The default value is 1.0. The change of the value is not necessary for normal calculations. However, the change of the value may speed up the convergence.

■ Convergence Tolerance

The convergence tolerance specifies the value that determines the convergence of iterations. The default is 1e-6.

■ Number of Iterations

If a iterative calculation does not converge, it is terminated when it reaches the maximum number of iterations. The number of iterations needs to be changed if a iteration calculation does not converge with the default value. The default value is 10000. If the number of iterations is 0, the direct method is automatically selected when the solver is executed.

■ Restart

The restart parameter needs to be specified when the GCR or GMRES method is used. The default value is normally used.

8-21-2-5. Parameters for Adaptive Method

The following parameters need to be set for the adaptive method. The adaptive method is available only when the GMG or Hybrid method is used. To change the number of divisions for mesh, change the following parameters. The default values can be used unless necessary.

8-21-2-6. Parameters for Multigrid Method

The following parameters need to be set for the multigrid method. The default settings can be used unless necessary.

■ Uniform Refinement Levels

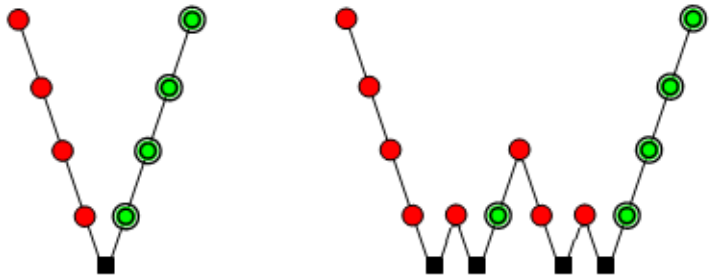
This parameter specifies the level number, for which the adaptive method is not used, among the steps used in the GMG method. For the steps specified for the level, the elements are subdivided uniformly without using the adaptive method.

■ Smoothing

This parameter specifies the number of smoothing operations at each step used for the GMG method.

■ Cycle Type

The V cycle and W cycle are available. The cycle type needs to be set when the GMG or Hybrid method is used.



V-cycle W-cycle

■ Maximum number of nodes in coarse mesh

This parameter needs to be set when the AMG method is used. The solver generates coarse mesh at the matrix level in the AMG method. The number of nodes for the most coarse mesh is specified.

8-22. Analysis Control: Control Output Items

▶▶ Reference

- See “Chapter 12. Setting Conditions of Electromagnetic Wave Frequency Response Analysis” “Analysis Control” of the JMAG-Studio User's Manual.
- See P.759 "8-26. Output Items of Electromagnetic Wave Frequency Response Analysis".

8-22-1. Function

The output items can be selected for analysis results.

■ Related items

- None

■ Settings

- Yes or no to output the magnetic field
- Yes or no to output the magnetic flux density
- Yes or no to output the current density
- Yes or no to output SAR (Specific Absorption Rate)
- Yes or no to output the analysis errors
- Specifying output method in calculating scattered field
  - Only the scattered field
  - The scattered field plus external field

8-23. User Subroutine

The following user subroutines can be used in a electromagnetic wave frequency response analysis.

8-23-1. Types of User Subroutine

■ *incusr*

incusr defines the electric field in the input port.

■ *extusr*

extusr defines the external field.

■ *plsusr_ini, plsusr, epsusr*

plsusr_ini, *plsusr*, and *epsusr* define the initial plasma density, plasma density, and permittivity.

8-23-2. Notes on Using User Subroutines

- Write user subroutines in Python or C++. C++ supports Microsoft Visual C++6.0 and Microsoft Visual C++.NET.
  - User subroutines for Python are in \solver\mod\wave\python\jmag.py under the JMAG installation directory.
  - User subroutines for C++ are in \solver\mod\wave\usrsub under the JMAG installation directory.

8-23-3. Using User Subroutines

- In using the user subroutines written in Python, recompiling modules is not necessary after the subroutines are modified since Python is a script language. For how to write the Python scripts, see the descriptions in "jmag-py" or the help file included with Python. The help file is available after Python is installed using "py152.exe".
  Change the descriptions of the part to be changed since both *incusr* and *extusr* functions are defined in "jmag.py."
  Since Python is a script language, execution speed drops when a complicated process is executed. In such cases, the execution speed can be improved by creating a DLL as a binary module and using it from Python. To create a DLL, Microsoft Visual C++6.0 or Microsoft Visual C++.NET is necessary.

- In using the user subroutines written in C++, recompiling the entire DLL is necessary if the C++ program file (*.cpp) is modified.

8-23-4. Creating the User Subroutine DLL

To create user subroutines, use either of the following development environments.

- Microsoft Visual C++
- Microsoft Visual C++.NET

MEMO If a file is opened in NET, the same file may not be opened in Visual C++. Be sure to check the development environment to be used, and then open a file. Be sure which development environment is used, and then open a file.

8-23-4-1. Using Microsoft Visual C++.NET

Important **Multiple files are copied in step 2. Please do not overwrite the existing files in case anything goes wrong.**

1. **Confirm that Microsoft Visual C++.NET is installed**
2. **Copy all of the files in the following directory to the JMAG installation directory.**

   `C: Program Files\JMAG-Studio\solver\mod\wave\NET\ins_dir`

   If there are files that have the same name, save the original files in another directory with different names.
3. **Double-click “usrsub.vcproj” to start Microsoft Visual C++.NET.**
4. **Open the C++ program file (*.cpp), rewrite the contents, and then save the file.**

   The name and location of the C++ program file (*.cpp) must not be changed.
   It is recommended that the original C++ program file (*.cpp) be renamed and saved.
   The “usrsub.cpp” file is the gateway to Python. Changes should be minimal.
5. **Create a DLL by building the “usersub” project using Solution Explorer.**

   After the build completes, the “usrsub.dll” file is created in the release directory.
6. **Place the DLL (usrsub.dll) created in step 5 in the directory where “jmag.py” is stored.**

▶▶ Reference
For the use of DLL, refer to the example of the use of P.745 "8-23-6. extusr".

8-23-4-2. Using Microsoft Visual C++

1. **Confirm that Microsoft Visual C++ is installed**
2. **Use version6.0: Intel version for Microsoft Visual C++.**

   Start Microsoft Visual C++ by double-clicking "urssub.dsw" in the following directory.

   `(JMAG installation directory)\solver\mod\wave\usrsub`

3. **If the necessary file (*.cpp) is not set for the project, "add the file to the project in the workspace".**

   Usually, this step is not necessary.

4. **Open the C++ program file (*.cpp), rewrite the contents, and then save it.**

   It is recommended that the original C++ program file (*.cpp) be renamed and saved. At this time, do not change the file location and file name. The "usrsub.cpp" file is the gateway to Python. Changes should be minimal.

5. **Select [Build] > [Rebuild], and then execute the recompiling of the DLL.**

   After the recompiling completes, the "usrsub.dll" file is created in the release directory.

6. **Place the DLL (usrsub.dll) created in step 5 in the directory where "jmag.py" is stored.**

▶▶ Reference

For details on how to use DLL from Python, refer to the sample to use extusr.

8-23-5. incusr+XX

8-23-5-1. Settings

- *incusr+XX* defines the electric field in the input port and returns the incident electric field (complex number and vector) for the specified modal number.
- The Function number is specified in the input port. Functions are called depending on the specified function number. The function name to be called is "*incusr*+ (function number)." For example, if "-10" is specified as a function number, "*incusr*10" is to be called.
- External variables that can be used are eps0 (vacuum permittivity), mu0 (vacuum permeability), c (velocity of light), jmag.frequency (current frequency), jmag.step (current step), and pi (circular constant).

8-23-5-2. Arguments

x, y, z	Coordinates (m)

8-23-5-3. Return Values

xv, yv, zv	Electric field vectors (V/m)

8-23-5-4. Example

In this example, the TE10 mode wave is specified for the modal number -10 that is incident on the plane in parallel with the ZX plane in a rectangular wave guide (30mm x 6mm) for the modal number -10.

■ **Program**

```
def incusr10(x,y,z):
    return te_mode(x,y,z)

def te_mode(x,y,z):
    n = 1
    m = 0
    a = 30.0e-3
    b = 6.0e-3
    h0 = 1.0
    cmu = mu0
    eps = eps0

    omega = 2.0*pi*jmag.frequency

    rk1=(n*pi/a)
    rk2=(m*pi/b)
    beta=sqrt(omega**2*eps*cmu-rk1**2-rk2**2)

    c1=-complex(0.0,1.0)*omega*cmu/(omega**2*cmu*eps+beta**2)
    c2=+complex(0.0,1.0)*omega*cmu/(omega**2*cmu*eps-beta**2)

    e1=-rk2*c1*h0*cos(rk1*x)*sin(rk2*z)
    e2=complex(0.0,0.0)
    e3=-rk1*c2*h0*sin(rk1*x)*cos(rk2*z)

    return (e1,e2,e3)
```

8-23-6. extusr

8-23-6-1. Settings

extusr defines the external field. It returns the electric field at the position of the coordinates passed from the program. The value of the external field is applied to the entire model. Therefore, complex operations decrease the execution speed.

8-23-6-2. Arguments

x, y, z	Coordinates (m)

8-23-6-3. Return Values

xv, yv, zv	Electric field vector (V/m)

8-23-6-4. Example 1

This example defines the plane wave of Z-polarized wave that has the Gaussian distribution toward the X-axis centered on x0 and propagates toward the Y-axis as an external field.

■ **Program**

```
def extusr(x,y,z):
    return gauss_beam2d(x,y,z)

def gauss_beam2d(x,y,z):
    j = complex(0.0,1.0)
    beta = 2 * pi * jmag.frequency / c
    w  = 1.0e-6
    x0 = 0.0
    xv = 0
    yv = 0
    zv = exp(-(x-x0)**2/w**2)*exp(j*beta*y)
    return (xv,yv,zv)
```

8-23-6-5. Example 2

In this example, DLL is called as an external field. To use DLL, remove the comment out on the third line of "jmag.py" to enable the "import usrsub" script.

■ **Program**

```
def extusr(x,y,z):
    omega = 2.0*pi*jmag.frequency
    (rvx,ivx,rvy,ivy,rvz,ivz) = usrsub.extusr(x,y,z,omega)
    vx = complex(rvx,ivx)
    vy = complex(rvy,ivy)
    vz = complex(rvz,ivz)
    return(vx,vy,vz)
```

8-23-7. plsusr_ini

8-23-7-1. Settings

plsusr_ini is used together with plasma conditions.

It returns the initial plasma density at the position of the coordinates passed from the program. This return value is applied to elements in a plasma calculation region specified with the plasma conditions.

When complicated calculations are conducted, the execution speed is reduced.

8-23-7-2. Arguments

x, y, z	Coordinate (m)

8-23-7-3. Return Values

pls	Plasma density ($1/m^3$)

8-23-7-4. Example 1

In this example, the plasma density 1.0e + 16 is set in the range of -1<x<1, -1<y<1, -1<z<1, and 1.0e + 15 is set in other ranges

■ Program

```
def plsusr_ini(x,y,z):
    return initial(x,y,z)

def initial(x,y,z):
    pls = 1.0e+15if x > -1.00:
    if x < 1.00:
           if y > -1.00:
                  if y < 1.00:
                         if z > -1.00:
                                if z < 1.00:
                                       pls = 1.0e+16
    return (pls)
```

8-23-7-5. Example 2

In this example, the DLL created with C++ is called. To use DLL, remove the comment out on the third line of "jmag.py" to enable the "import usrsub" script. As for the codes of C++, refer to the samples in the installation directory.

■ **Program**

```
def plsusr_ini(x,y,z):
    return initial(x,y,z)

def initial(x,y,z):
    (pls) = usrsub.plsini(x,y,z)
    return (pls)
```

8-23-8. plsusr

8-23-8-1. Settings

plsusr is used together with plasma conditions.

It returns the next most iterative plasma density for the electric field value and plasma density passed from the program. This return value is applied to elements in a plasma calculation region specified with the plasma conditions.

When complicated calculations are conducted, the execution speed is reduced.

8-23-8-2. Arguments

M [1]	Element ID
IEL [1]	Number of nodes in element
NOD [8]	Node ID within an element
XYZ [3*8]	Coordinates of each node (In the order of x, y, z, x, y, z) (m)
E [6]	Electric field value Ex real part, Ex imaginary part, Ey real part, Ey imaginary part, Ez real part, Ez imaginary part) (Vm)
Plold [1]	Plasma density ($1/m^3$)

These are the information of elements in a plasma calculation region specified with the plasma conditions. The subscripts 1 through 6 are used to pass the information of peripheral elements (up to 6 elements).

8-23-8-3. Return Values

nnew	Plasma density (next step) ($1/m^3$)

8-23-8-4. Example 1

In this example, the plasma density of the next step is determined by using the electric field value and plasma density of the specified elements.

As an example, the model in the "Reference data" is provided.

■ Program

```
def plsusr(*obj):
    M = obj[0]
    IEL = obj[1]
    NOD = obj[2]
    XYZ = obj[3]
    E = obj[4]
    plold = obj[5]

    M1 = obj[6]
    IEL1 = obj[7]
    NOD1 = obj[8]
    XYZ1 = obj[9]
    E1 = obj[10]
    plold1 = obj[11]

    M2 = obj[12]
    IEL2 = obj[13]
    NOD2 = obj[14]
    XYZ2 = obj[15]
    E2 = obj[16]
    plold2 = obj[17]

    M3 = obj[18]
    IEL3 = obj[19]
    NOD3 = obj[20]
    XYZ3 = obj[21]
    E3 = obj[22]
    plold3 = obj[23]

    M4 = obj[24]
    IEL4 = obj[25]
    NOD4 = obj[26]
    XYZ4 = obj[27]
    E4 = obj[28]
    plold4 = obj[29]

    M5 = obj[30]
    IEL5 = obj[31]
    NOD5 = obj[32]
    XYZ5 = obj[33]
    E5 = obj[34]
    plold5 = obj[35]
```

```
    M6 = obj[36]
    IEL6 = obj[37]
    NOD6 = obj[38]
    XYZ6 = obj[39]
    E6 = obj[40]
    plold6 = obj[41]

    return plasma(M,IEL,NOD,XYZ,E,plold,

M1,IEL1,NOD1,XYZ1,E1,plold1,M2,IEL2,NOD2,XYZ2,E2,plold2,
         M3,IEL3,NOD3,XYZ3,E3,plold3,
M4,IEL4,NOD4,XYZ4,E4,plold4,
         M5,IEL5,NOD5,XYZ5,E5,plold5,
M6,IEL6,NOD6,XYZ6,E6,plold6)


def plasma(M,IEL,NOD,XYZ,E,plold,
M1,IEL1,NOD1,XYZ1,E1,plold1, M2,IEL2,NOD2,XYZ2,E2,plold2,
M3,IEL3,NOD3,XYZ3,E3,plold3, M4,IEL4,NOD4,XYZ4,E4,plold4,
M5,IEL5,NOD5,XYZ5,E5,plold5, M6,IEL6,NOD6,XYZ6,E6,plold6):
    xr = E[0]
    xi = E[1]
    yr = E[2]
    yi = E[3]
    zr = E[4]
    zi = E[5]

    ex = complex(xr,xi)
    ey = complex(yr,yi)
    ez = complex(zr,zi)

    exa = abs(ex)
    eya = abs(ey)
    eza = abs(ez)

    ele = math.sqrt(exa*exa + eya*eya + eza*eza)

    em = 1500
    eb = 10000
    ganma = 3.0e+12
    nmin = 1.0e+16
    nth = 1.0e+15

    if ele <= em:
                nnew = 0.0
            else:
        if ele < eb:
                    if plold[0] > nth:
                        nnew = ganma * (ele - em) + nmin
                else:
                    nnew = 0.0
            else:
                nnew = ganma * (ele - em) + nmin
```

```
    return (nnew)
```

MEMO Reference data
Simulation and development of optimized microwave plasma reactors for diamond deposition, Surface&Coatings Technology 116-119(1999)

8-23-8-5. Example 2

In this case, the DLL created with C++ is called. To use DLL, remove the comment out on the third line of "jmag.py" to enable the "import usrsub" script. (Only plasma is inserted. *plsusr* is the same as Example 1.) As for the codes of C++, refer to the samples in the installation directory.

■ Program

```
def plasma(M,IEL,NOD,XYZ,E,plold,
M1,IEL1,NOD1,XYZ1,E1,plold1, M2,IEL2,NOD2,XYZ2,E2,plold2,
M3,IEL3,NOD3,XYZ3,E3,plold3, M4,IEL4,NOD4,XYZ4,E4,plold4,
M5,IEL5,NOD5,XYZ5,E5,plold5, M6,IEL6,NOD6,XYZ6,E6,plold6):

    (nnew) = usrsub.plsusr(M,IEL,NOD,XYZ,E,plold,
    M1,IEL1,NOD1,XYZ1,E1,plold1, M2,IEL2,NOD2,XYZ2,E2,plold2,
    M3,IEL3,NOD3,XYZ3,E3,plold3, M4,IEL4,NOD4,XYZ4,E4,plold4,
    M5,IEL5,NOD5,XYZ5,E5,plold5, M6,IEL6,NOD6,XYZ6,E6,plold6)

    return (nnew)
```

8-23-9. epsusr

8-23-9-1. Settings

epsusr is used together with plasma conditions.

It returns the relative permittivity for the plasma density, center coordinate of elements, and electric field passed from the program. This return value is applied to elements in a plasma calculation region specified with the plasma conditions. When complicated calculations are conducted, the execution speed is reduced.

8-23-9-2. Arguments

pls	Plasma Density (1/m3)
x	X-coordinate of the element's center
y	Y-coordinate of the element's center
z	Z-coordinate of the element's center
exr	Electric field X component (real)
exi	Electric field X component (imaginary)
eyr	Electric field Y component (real)
eyi	Electric field Y component (imaginary)
ezr	Electric field Z component (real)
ezi	Electric field Z component (imaginary)

8-23-9-3. Return Values

epsilon	relative permittivity (complex number)

8-23-9-4. Example 1

In this example, a model with elementary electric charge is used. As an example, the model in the "Reference data" is provided.

■ **Program**

```
def epsusr(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi):
                            return
epsilon(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi)

import math
def epsilon(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi):
#       use Python.
         freq = 2.45e+9
         pi = 3.14159
         elec = 1.60e-19
         me = 9.1093897e-31
     a = 1.0e+10
         pdevt = 3.6
         nyu = a * pdevt
         ep0 = 8.854e-12
         omegap = math.sqrt(pls * elec*elec / me / ep0)

         omega = 2.0*pi*freq
         valr = ep0*(1-omegap*omegap/(omega*omega+nyu*nyu))
         vali = -ep0*nyu*omegap*omegap/
(omega*(omega*omega+nyu*nyu))

         valr_r = valr / ep0
         vali_r = vali / ep0

    eps = complex(valr_r,vali_r)
    return (eps)
```

8-23-9-5. Example 2

In this example, the DLL created with C++ is called. To use DLL, remove the comment out on the third line of "jmag.py" to enable the "import usrsub" script.

■ Program

```
def epsusr(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi):
     return epsilon(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi)

import math
def epsilon(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi):
     (valr_r,vali_r) =
usrsub.epsusr(pls,x,y,z,exr,exi,eyr,eyi,ezr,ezi)

     eps = complex(valr_r,vali_r)
     return (eps)
```

MEMO Reference data

Simulation and development of optimized microwave plasma reactors for diamond deposition, Surface&Coatings Technology 116-119(1999)

MEMO As of October 2007, part of the specification of *epsusr* is changed. Due to this change, "epsusr" is no longer be available for the "jmag.py" files and DLL files in C++.

8-24. Far Field Calculation

8-24-1. Function

The Calculate Far Field tool calculates the electromagnetic field in regions that are sufficiently far from the finite element analysis region by using the equivalent surface current on the specified surface of an analysis model. The calculation is performed using the Calculate Far Field tool (CALFAR).

▶▶ Reference
For more information, see P.947 "12-6. Calculate Far Field (JMAG-Studio)".

■ Related items

- Analysis control condition (control of output items)
- External field condition

■ Targets

- Analysis model

■ Settings

- Equivalent surface current file
- Calculation point

■ Output items

- None

8-24-2. Important Notes Regarding the Use of This Function

- The Calculate Far Field tool (CALFAR) is a program that is separated from the WV module. To start this tool, select [Tools] > [Calculate Far Field] from the menu bar.
- The equivalent surface current file is exported when the equivalent current calculation condition is set. The name of the equivalent surface current file is to be “xx.srfcur” consisting of the input data name “xx” and extension “srfcur.”
- The far field calculation is performed on the sphere with radius specified centering on the origin of the global coordinate system where the finite element model is defined.
- The calculation point is determined by specifying the number of divisions toward the latitude and longitude in the polar coordinate system in which the Z-axis in the global coordinate system is defined as the polar axis.
- When using the external field as input, the far field for scattered waves is calculated. The external field is excluded.

8-25. Equivalent Current Calculation

8-25-1. Function

The equivalent current calculation condition specifies a face to calculate where the equivalent surface current used for a far field calculation flows.

■ Related items

- Far field calculation tool

■ Targets

- Element face

■ Settings

- None

■ Output items

- Equivalent surface current

8-25-2. Important Notes Regarding the Use of This Function

- When this condition is set, an equivalent surface current file (*.srfcur) is exported.

8-26. Output Items of Electromagnetic Wave Frequency Response Analysis

The electromagnetic wave frequency response analysis module (WV) can display the following items as analysis results.

Output item	Unit	Type	Object	Note
Magnetic field	A/m	Vector	Solid	
Magnetic flux density	T	Vector	Solid	
Electric field	V/m	Vector	Solid	
Joule loss	W	Step	-	
Joule loss density	W/m^3	Scalar	Solid	
SAR (Specific Absorption Rate)	W/kg	Scalar	Solid	
Transmitted power	W	Step	-	
Current	A	Step	-	
Current density	A/m^2	Vector	Solid	
Voltage	V	Step	-	This item is exported when the voltage calculation condition is specified.
Hysteresis loss	W	Step	-	
Hysteresis loss density	W/m^3	Scalar	Solid	
Plasma density	$1/m^3$	Scalar	Solid	This item is exported when plasma condition is specified.ma condition is specified.
Quality factor	-	Step	-	This item is exported when plasma condition is specified.
Analysis error	-	Scalar	Solid	

Chapter 9. Mesh

This chapter describes the mesh generation.

- Adaptive Mesh Control (Magnetic Field Analysis) (762 page)
- Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (767 page) * JMAG-Designer Only

▶▶ Reference

For more information, see Chapter 15, “Generating Mesh on Geometry Editor (Manual)” and Chapter 36, “Generating Mesh (Automatic)” in the JMAG-Designer User's Manual.

9-1. Adaptive Mesh Control (Magnetic Field Analysis)

▶▶ Reference

- For more information, see "Adaptive Mesh Control" in Chapter 9, "Setting Conditions of Magnetic Field Analysis" in the JMAG-Studio User's Manual.
- For more information, see "Adaptive Mesh" in Chapter 36, "Generating Mesh (Automatic)" in the JMAG-Designer User's Manual.

9-1-1. Function

The adaptive mesh function is used to generate an optimum mesh by (1) calculating all steps, (2) evaluating an analytical error, and (3) refining the mesh to perform a highly accurate magnetic field analysis.

■ Related items

- Symmetry boundary
- Motion

■ Settings

- Acceptable error range
- Number of stages
- Number of remesh per stage
- Flag for considering nodal force (2D)
- Flag for saving result at each stage of adaptive mesh generation

■ Available modules

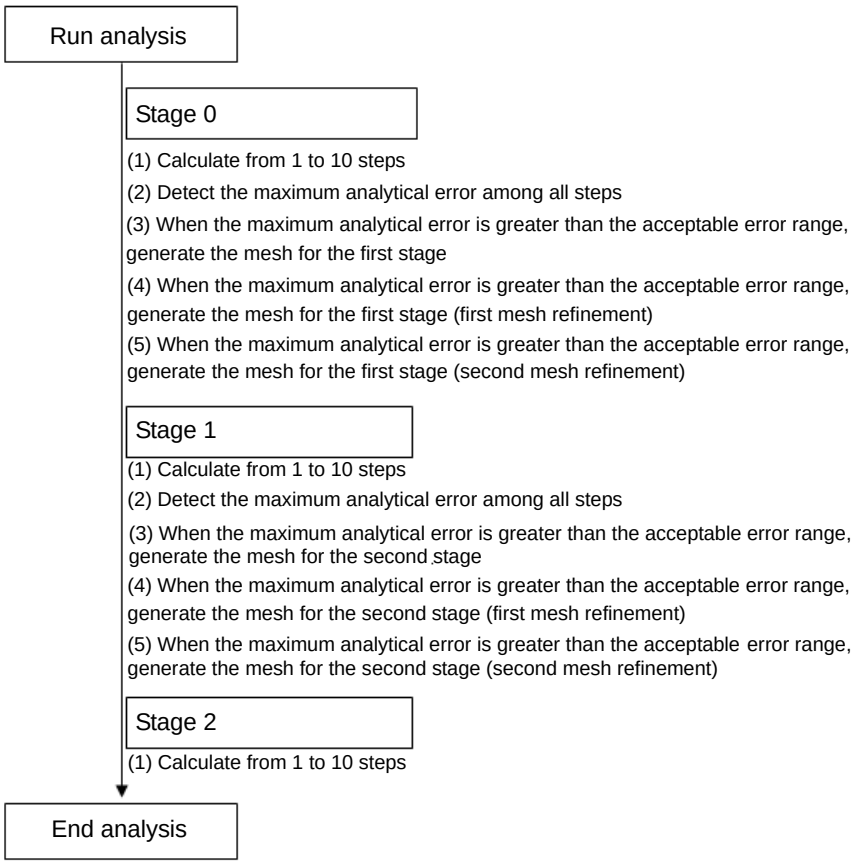
- All of the modules of magnetic field analysis

9-1-2. Important Notes Regarding the Use of This Function

- The adaptive mesh function can be used for the following regions and elements.
  - 2D analysis -> region
  - 3D analysis -> mesh (hexahedral element, pentahedron element, and tetrahedral element) or CAD data

  MEMO If a mesh is already generated in the model before the adaptive mesh function is used, the mesh is called “initial mesh.” By generating an initial mesh in advance, the size of the mesh to be generated by the adaptive mesh function can be controlled to some extent.

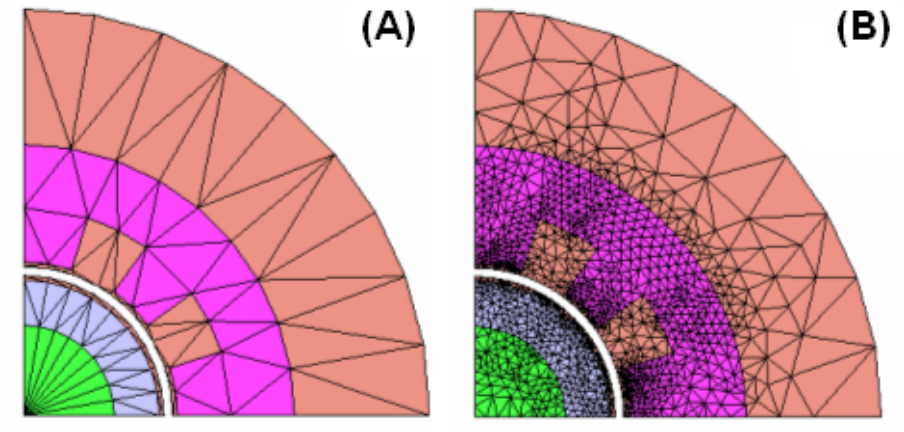
- In 2D analyses, all elements generated using the adaptive mesh function are a triangular element regardless of the initial mesh. In 3D analyses, are element are a quadrilateral element.
- The analysis steps using the adaptive mesh function is as follows:
  For example:
  - For the step control settings, [Steps] is set to 10.
  - For the adaptive mesh control settings, [Maximum Number of Stages] is set to 2, and [Remesh per Stage] is set to 2.


Run analysis
Stage 0
(1) Calculate from 1 to 10 steps
(2) Detect the maximum analytical error among all steps
(3) When the maximum analytical error is greater than the acceptable error range, generate the mesh for the first stage
(4) When the maximum analytical error is greater than the acceptable error range, generate the mesh for the first stage (first mesh refinement)
(5) When the maximum analytical error is greater than the acceptable error range, generate the mesh for the first stage (second mesh refinement)
Stage 1
(1) Calculate from 1 to 10 steps
(2) Detect the maximum analytical error among all steps
(3) When the maximum analytical error is greater than the acceptable error range, generate the mesh for the second stage
(4) When the maximum analytical error is greater than the acceptable error range, generate the mesh for the second stage (first mesh refinement)
(5) When the maximum analytical error is greater than the acceptable error range, generate the mesh for the second stage (second mesh refinement)
Stage 2
(1) Calculate from 1 to 10 steps
End analysis

- In 3D analyses, the adaptive mesh function cannot be used with the patch mesh function.
- The adaptive mesh function can be used with the restart function. To restart the mesh generation, the mesh data file (*.msv), which is exported after the analysis is complete, needs to be specified.
- If the analysis is named "XXXX," the following files are created at the end of calculation:
  - XXXX.jcf
    Input file (model data)
  - XXXX.jplot
    Result file of JMAG-Designer
  - XXXX.plot
    Result file of JMAG-Studio
  - XXXX_input.jcf
    Input file including mesh data created using the adaptive mesh function
    This file is used as an input file for the following:
    - Electromagnetic force condition for a structural analysis
    - Tool: Calculate Iron Loss
    - Tool: Force for Structural Analysis

  MEMO If the adaptive mesh function is used for the model with initial mesh, "*_input.jcf" is not exported.
- The element size can be specified at stage 0 for refining the mesh for the first time. The mesh is refined based on the specified element size.
  - In JMAG-Studio, the element size specified for regions, solids, and air regions in the [Automatic Generation] dialog box is used.

- In JMAG-Designer, the element size specified for parts, faces, edges, and vertices is used for refining the mesh. The element size for an air region can be specified in the [Mesh Properties] dialog box.



(a) Plot file for the mesh refinement for the first time
(b) Plot file for the mesh refinement for the fifth time

9-1-2-1. Acceptable Error Range and Maximum Number of Stages

- An analytical error is a value obtained by evaluating the continuity of magnetic flux between adjacent elements based on the stored energy of the entire model. The unit of measurement is (%).
- In the analysis using the adaptive mesh function, the maximum analytical error is selected from the analytical errors of multiple steps, and then the maximum analytical error is compared with the acceptable error range.
- The default acceptable error range is 0.2%, and the default maximum number of stages is 1.
- If an analytical error is greater than the specified acceptable error range, the analysis ends as soon as the specified number of stages is reached.
  For example, given that the analysis is run with the acceptable error range of 0.1% and the number of stages of 2, and some elements have analytical error of 1% at the end of the second stage. In this case, the analytical error of 1% is greater than the acceptable error range of 0.1%, so the calculation for the third stage is necessary; however, the analysis ends because the specified number of stages is reached.
- Even if the number of stages does not reach the limit, if the analytical error of all elements is less than the acceptable error range, the analysis ends.
  For example, given that the analysis is run with the acceptable error range of 0.2% and the number of stages of 2, and the maximum analytical error is 0.1% at the end of the first stage. In this case, the maximum analytical error of 0.1% is smaller than the acceptable error range, so the calculation for the second stage is not necessary.

9-1-2-2. Remesh per Stage

- This setting specifies the number of times to refine the mesh per stage. The default value is 2.
- A mesh is always generated at least once for each stage. The first mesh generation is not included in the number of remesh specified in [Remesh per Stage].
- Areas that are predicted to have large analytical errors in the previous mesh are divided even more when remeshing is run.
- The default value is recommended for the number of remesh per stage. The larger the number of remesh is, the more time is required to generate the mesh.

9-1-2-3. Calculate Nodal Force Accurately

- This function is used to obtain a stable calculation result for electromagnetic force or torque by subdividing the elements where variation in nodal force is intense and distributing the nodal force uniformly.
- Elements where variation in nodal force is intense should be treated as elements with larger analytical error.
- If variation in nodal force is considered as an analytical error, the [Nodal Force] check box needs to be selected for the [Output Control] settings.
- With this setting, more calculation time is required since nodal force data needs to be extracted from the result file (*.jplot, *.plot). Also, when variation in nodal force is intense, elements are continuously determined to have a larger analytical error and the calculation may be iterated until it reaches the maximum number of stages.

9-1-2-4. Save Result at Each Stage of Adaptive Mesh Generation

- When the [Save result at each stage if adaptive mesh generation] is selected, a plot file at each stage is exported.

9-2. Add a Layer of Air in the Computational Domain Force and Torque (Magnetic Field Analysis) (JMAG-Designer)

▶▶ Reference

For more information, see Chapter 36, “Generating Mesh (Automatic)” in the JMAG-Designer User's Manual.

9-2-1. Function

An air layer is added to the setting area of the force/torque condition, when using the (FEM+BEM) method with electromagnetic force/torque condition.

■ Related items

- Force/torque calculation
- Calculation method

■ Setting

- Air layer thickness
- Air layers

■ Modules that can be used with this function

- FEM+BEM is selected as the calculation method for a 2D analysis from the selection of magnetic field analysis models.

9-2-2. Important Notes Regarding This Function

- [Add a layer of air in the computational domain force and torque] is specified when using the solution method (FEM+BEM) with the electromagnetic force/torque condition (nodal force method).
  A mesh does not need to be created in air region when the solution method (FEM+BEM) is selected. The force in each node positioned on the outside edge of the setting area cannot be calculated properly, if an air layer does not exist in the setting area of the electromagnetic force/torque calculation condition (nodal force method). The calculation for the electromagnetic force/torque is calculated properly, regardless if the solution method (FEM+BEM) is used with the electromagnetic force/torque calculation condition (nodal force method). This occurs when air layer is automatically added on the outer area of a target setting for an electromagnetic force/torque calculation condition during analysis, by using [Add a layer of air in the computational domain force and torque].

Chapter 10. Coupled Analysis

In JMAG, an analysis run by combining multiple solvers (MAG (TR, DP, FQ, ST), HT, EL, CH, DS) is called a “coupled analysis”.

This chapter describes the types of coupled analysis and important points to consider.

▶▶ Reference

For more information about coupled analysis for Abaqus, see "Chapter 11. Coupled Analysis Using Abaqus" on page 823.

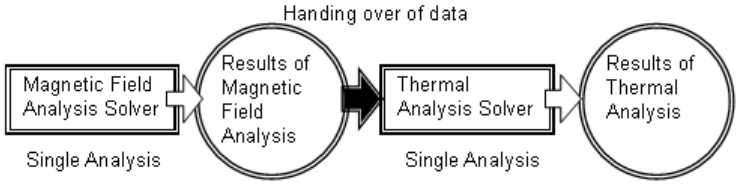
10-1. Types of Coupled Analysis

In coupled analysis, by utilizing multiple solvers, it is possible to make analysis that puts other fields into consideration. For instance, it is possible to consider heat source in magnetic fields.

In the coupled analysis conducted by JMAG, there are roughly the following two types.

■ One-way coupled analysis

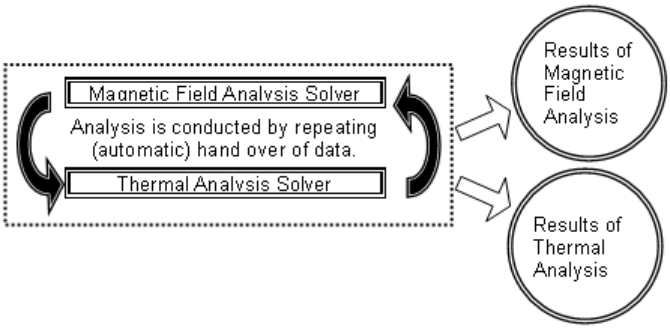
This is method in which analysis is conducted in one of the solvers, and its results are utilized in the other solver. In the following case, the thermal analysis is run based on the results of the magnetic field analysis. The thermal analysis results will not affect the magnetic field analysis.



Example: Coupling in one direction (MAG and HT)

■ Two-way coupled analysis

This is a method in which analyses are executed simultaneously by conducting the hand over of data mutually. In the following case, the magnetic field analysis and thermal analysis are run alternately. Each time the magnetic field analysis is run, the thermal analysis results are taken into account, and vice versa.


Magnetic Field Analysis Solver
Analysis is conducted by repeating (automatic) hand over of data.
Thermal Analysis Solver
Results of Magnetic Field Analysis
Results of Thermal Analysis

Example: Coupling in both directions (MAG and HT)

10-2. Solver Combinations for Coupled Analysis

Coupled analysis is possible for the following combinations of solvers. Read this table with reference to the left column.

	TR	DP	ST	FQ	LS	HT	EL-ST	EL-FQ	EL-CUR	CH	DS-ST	DS-EIG	DS-FQ
TR		-	-	-	-	=> <=>	-	-	-	-	=>	-	=>
DP	-		-	-	-	=> <=>	-	-	-	-	=>	-	=>
ST	-	-		-	-	=>	-	-	-	-	=>	-	-
FQ	-	-	-		-	=> <=>	-	-	-	-	=>	-	=>
LS	-	-	-	-		=>	-	-	-	-	-	-	-
HT	=> <=>	=> <=>	-	=> <=>	-		-	-	=> <=>	-	=>	-	-
EL-ST	-	-	-	-	-	-		-	-	-	-	-	-
EL-FQ	-	-	-	-	-	=> <=>	-		-	-	-	-	-
EL-CUR	-	-	-	-	-	=> <=>	-	-		-	-	-	-
CH	-	-	-	-	-	-	-	-	-		-	-	-
DS-ST	=>	=>	=>	=>	-	-	-	-	-	-		-	-
DS-EIG	-	-	-	-	-	-	-	-	-	-	-		-
DS-FQ	-	-	-	-	-	-	-	-	-	-	-	-	

<=>: Coupling in both directions is possible

=>: Coupling in one direction is possible (from the solver on the left most column to the solver on the top line)

-: Coupling is not possible

Solver abbreviations are as follows:

- TR: Magnetic field analysis (transient response analysis) *3D
- DP: Magnetic field analysis (transient response analysis) *2D, axisymmetric
- ST: Magnetic field analysis (static analysis) *3D, 2D, axisymmetric
- FQ: Magnetic field analysis (frequency response analysis) *3D, 2D, axisymmetric
- LS: Iron loss calculation *3D, 2D
- HT: Thermal analysis *3D

- EL-ST: Electric field analysis (static analysis) *3D
- EL-FQ: Electric field analysis (frequency response analysis) *3D
- EL-CUR: Current distribution analysis *3D
- CH: charge distribution analysis *3D
- DS-ST: Structural analysis (static analysis) *3D, 2D
- DS-EIG: Structural analysis (eigenmode analysis) *3D, 2D
- DS-FQ: Structural analysis (frequency analysis) *3D, 2D

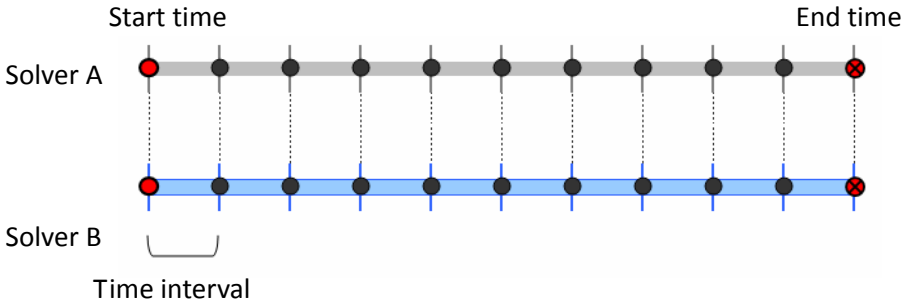
MEMO The two-way coupled analysis for magnetic field analysis and structural analysis in JMAG is not supported. However, a two-way coupled analysis for a magnetic field analysis in JMAG and a structural analysis in Abaqus is supported.

▶▶ Reference

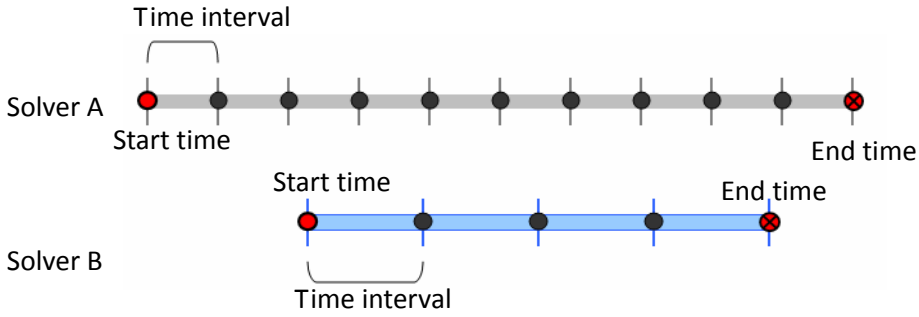
For more information, see "Chapter 11. Coupled Analysis Using Abaqus" on page 823.

10-3. Time Interval Settings for Two-way Coupled Analysis (JMAG-Designer)

A solver with different start time, end time, time interval, and number of steps, can be specified in a two-way Coupled Analysis.



A regular pattern for a coupled analysis
(Start time, end time, and time interval, and number of steps match with A and B)



Example of a pattern set with time intervals for each solver
(Start time, end time, and time interval, and number of steps differ with A and B)

■ Types of supported coupled analysis

- Magnetic field analysis (transient response analysis) (average loss) <-> thermal analysis (783 page)
- Magnetic field analysis (frequency response analysis) <-> thermal analysis (795 page)

■ Important Notes Regarding the Use of This Function

- Time interval settings are specified in the [Coupled Files] dialog box of JMAG-Scheduler.

  ▶▶ Reference
  For more information about JMAG-Scheduler, see "Chapter 40. Running the Analysis," in the JMAG-Designer User's Manual.

- Input file (*.jcf) for thermal analysis cannot specify time interval.Specify an input file coupling with a thermal analysis (magnetic field analysis (transient response analysis)). An analysis start time that is before the start of the thermal analysis cannot be coupled with a thermal analysis.
- Specify the analysis start time to be the same or later than the thermal analysis.
- Regular intervals are supported. Irregular time intervals are not supported.

■ Setting

The settings for the time interval can be specified from the following items.

- Start Time to Apply
  Specifies the time for the first step.
- End Time to Apply
  Specified the time for the last step.
- Divisions of Time
  Specifies the number of divisions in an area determined from an analysis time interval and analysis end time. The time interval and the number of steps are set with this specification.
- Skip End Time
  Select this check box to prevent calculation from running in end time. For more information, see "Using flags in [Skip End Time]" on page 777 .

Important **The time interval set in the input file for the solver which is not set with time intervals are used. For example, the analysis start time, analysis end time, and number of divisions can be set for a magnetic field analysis, when running a coupled magnetic field analysis and thermal analysis. The analysis start time, analysis end time and the number of divisions for a thermal analysis, is also applied to the magnetic field analysis when the values above is not specified in a magnetic field analysis.**

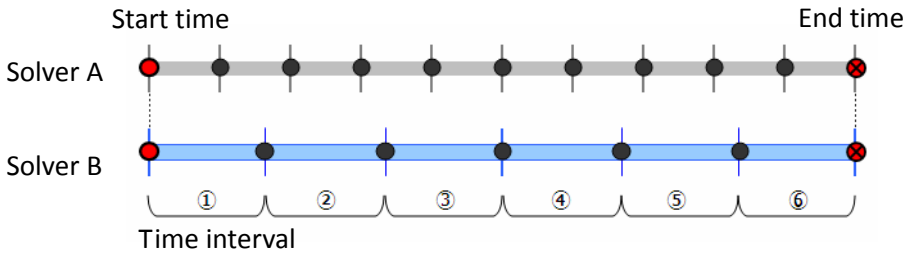
■ Example of pattern settings for time interval

The following example describes Solver B specifying settings for time interval.

■ Example 1

Pattern specifying [Divisions] for solver B. Settings for [Start time to apply] and [End time to apply] are not specified.

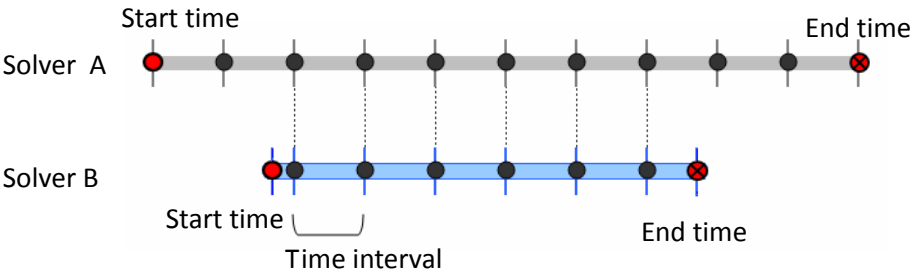
Values in solver A are used when settings for [Start time to apply] and [End time to apply] are not specified. In this example, '6' is specified in [Divisions] in which '7' is specified as the number of steps.



■ Example 2

Pattern when [Start time to apply] for solver B is set slower than solver A, and [End time to apply] for solver A is set faster. [Divisions] is not specified.

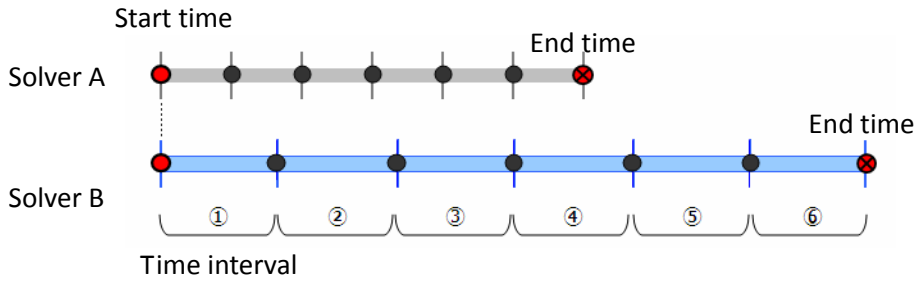
[Divisions] is not specified, in which the value in solver A is used for time interval. However, the number of steps for solver A and B is different when [Start time to apply] and [End time to apply] is specified.



■ Example 3

Pattern when [End time to apply] for solver B is slower than solver A, and [Divisions] is set. [Start time to apply] is not specified.

Values in solver A is used when [Start time to apply] is not set. In this example, '6' is specified in [Divisions] in which '7' is specified as the number of steps.

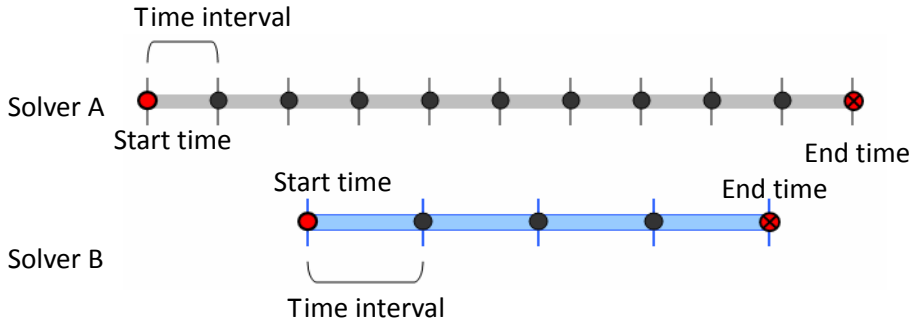


Important **A faster time cannot be set for solver B in [Start time to apply] than solver A.**

■ Example 4

Pattern set with [Start time to apply], [End time to apply], and [Divisions] is set for solver B.

The settings for [Start time to apply], [End time to apply], time interval, and number of steps differ in settings for solver A and B.

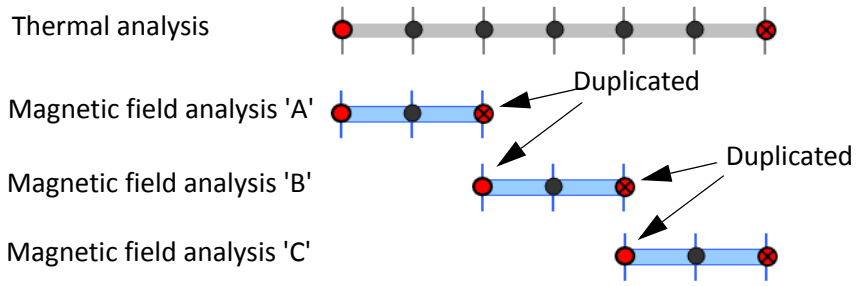


■ Using flags in [Skip End Time]

The calculation in the end time is not run when this flag is selected.

This flag is used to prevent calculations from running when multiple analyses are overlapping with the same time interval, and is specified to a single JCF file.

For example, JCF files 'A,' 'B,' and 'C' used for a coupled magnetic field analysis are specified for a single JCF file of a thermal analysis in the following diagram. The end time for magnetic field analysis 'A' and the start time for magnetic field analysis 'B,' and the end time for the magnetic field analysis 'B' and the start time for magnetic field analysis 'C' all match. Therefore, the calculation runs with duplicated time since the files are separate.



Therefore, the calculation is not run for the end time in magnetic field analysis 'A' and 'B', when the [Skip End Time] check box is selected for magnetic field analysis A and B.

The [Skip End Time] check box cannot be selected for the last analysis (magnetic field analysis C), and the calculation cannot be set to be skipped.

10-4. Coupling Magnetic Field Analysis and Thermal Analysis in One Direction

■ Necessary solvers

- Magnetic field analysis (one of the following)
  - Static magnetic field analysis (ST)
  - Transient response magnetic field analysis (TR, DP)
  - Frequency response magnetic field analysis (FQ)
  - Iron loss calculation (LS) *Only in the case of "LS => HT"
- Thermal analysis (HT)

■ Types of analysis

- Static magnetic field analysis => Thermal analysis
- Transient response magnetic field analysis => Thermal analysis
- Frequency response magnetic field analysis => Thermal analysis
- Iron loss calculation => Thermal analysis

- Thermal analysis => Magnetic field analysis (static analysis)
- Thermal analysis => Magnetic field analysis (frequency response analysis)
- Thermal analysis => Magnetic field analysis (transient response analysis)

10-4-1. Magnetic Field Analysis, Iron Loss Calculation => Thermal Analysis

After calculating joule loss density with the magnetic field analysis, use it as a heat source, and run steady-state or transient-state analysis independently with the thermal analysis solver.

10-4-1-1. Data transferred between solver programs and where it is received

- Magnetic field analysis Joule loss => Thermal analysis, Heat source condition
- Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Magnetic field analysis Joule loss + Iron loss analysis, Iron loss => Thermal analysis, Heat source condition

10-4-1-2. Important Notes Regarding the Use of This Function

- Temperature dependence of electrical conductivity and relative permeability cannot be taken into account.
- Data needs to be created separately for the thermal analysis and for the magnetic field analysis.

10-4-1-3. Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

■ Study properties / Analysis control

- Turn off the flag for coupling with thermal analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Clear the [Coupling with Thermal] check box in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Joule Loss Density] check box.
  - Select the [Joule Loss Density] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Joule Loss Density] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.

■ Condition

- The heat source condition needs to be set when the loss in an FEM coil and resistance component is used for a thermal analysis.
  - If this condition is set for a partial model, the conversion factor needs to be set for both the FEM model and the circuit. The conversion factor can be specified with the analysis control condition in JMAG-Studio and with the study properties in JMAG-Designer.

10-4-1-4. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Normal Run] radio button under [Analysis Type] in the [Analysis Control] dialog box for JMAG-Studio.

■ Condition

- Set the heat source condition and select [One Way] as the coupling type. Specify to refer a result file for a magnetic field analysis or an iron loss analysis.
  - If the [Use Average Joule Loss] check box is not selected when referencing the result file of a transient analysis with multiple steps, the joule loss for the last step is used as the heat source.
  - When a partial model is used, if the heat source type is specified to the total amount for the heat source condition, the loss (total amount of heat source) obtained with magnetic field analysis needs to be converted using one of the following methods.
    - Specify the partial model condition (recommended)
    - Specify the loss scale for the heat source condition

10-4-2. Thermal Analysis => Magnetic Field Analysis (JMAG-Designer)

Run an independent analysis of a magnetic field analysis solver by referring to a calculated temperature in a thermal analysis.

10-4-2-1. Data transferred between solver programs and where it is received

- Thermal analysis, Temperature => Magnetic field analysis, Temperature dependent material, Temperature distribution condition

10-4-2-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a coupled analysis is created separately per solver.

10-4-2-3. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for a study property.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties].

10-4-2-4. Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

■ Material

- Creates a material with temperature dependent characteristic that is set to a part.
  - Set a temperature - electric conductivity curve as an electric property for material settings when using temperature dependency for electric conductivity.
  - Set a curve for temperature - relative permeability when setting the materials to use the temperature dependency of relative permeability.

- Set a curve for temperature - magnetic susceptibility when using the temperature dependency for a magnetic property defined by a B-H curve.

  ▶▶ Reference

  For more information, see "2-3. Magnetizing Properties of a Soft Magnetic Material" on page 20 and "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)" on page 33.

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency. Select [One Way] as the coupling type and specify the result file (*.jplot, *.plot) for a thermal to be referenced.

10-5. Coupling Magnetic Field Analysis and Thermal Analysis in Both Directions

■ Necessary solvers

- Magnetic field analysis (one of the following)
  - Frequency response magnetic field analysis (FQ)
  - Transient response magnetic field analysis (TR, DP)
- Thermal analysis (HT)

■ Types of analysis

- Transient response magnetic field analysis (in which a loss-averaged flag is used) <=> Thermal analysis
- Transient response magnetic field analysis <=> Thermal analysis
- Frequency response magnetic field analysis <=> Thermal analysis

10-5-1. Magnetic Field Analysis (Transient) (Loss Averaging Flag) <=> Thermal Analysis

The average loss for time (step) in a magnetic field analysis is calculated and transferred to a thermal analysis.

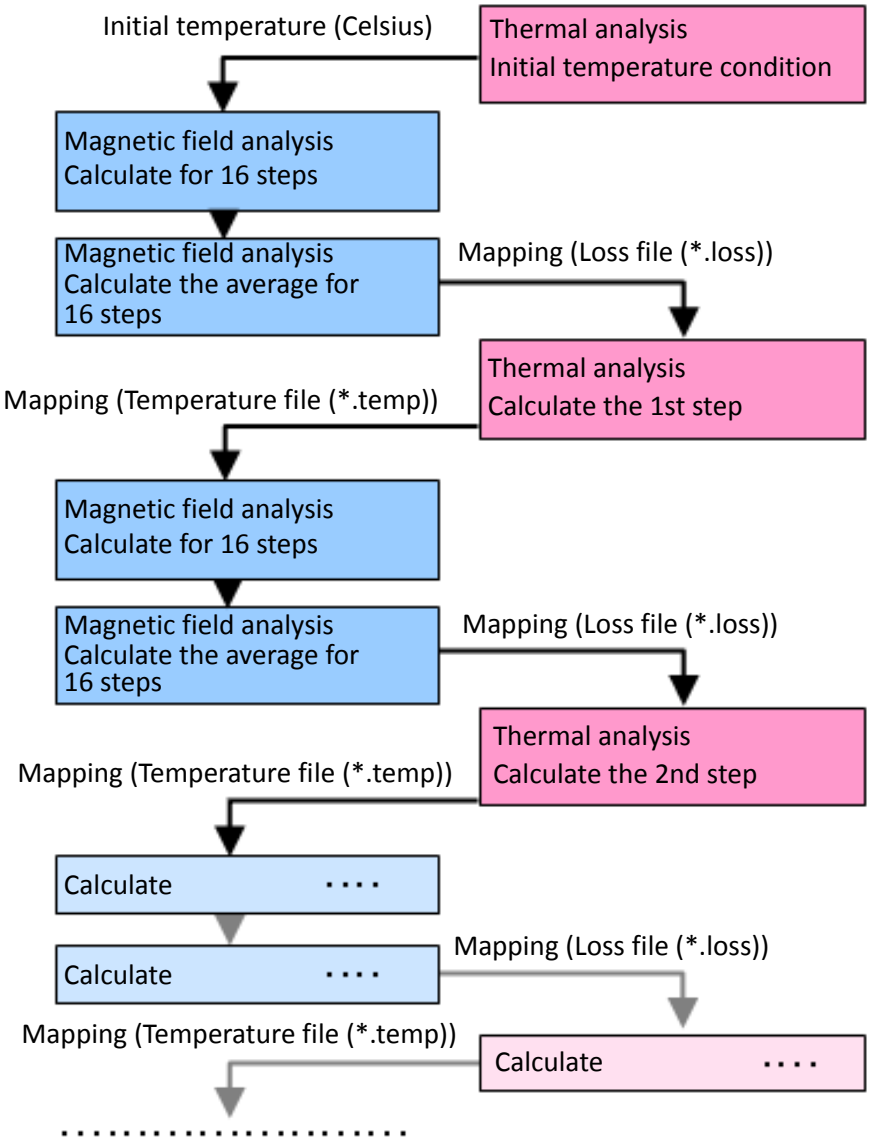
An iron loss analysis is run after a magnetic field analysis when an iron loss condition is set in a magnetic field analysis.

The average loss calculated from a magnetic field analysis added with an iron loss calculated from an iron loss calculation is transferred to the thermal analysis.

■ Processing procedures

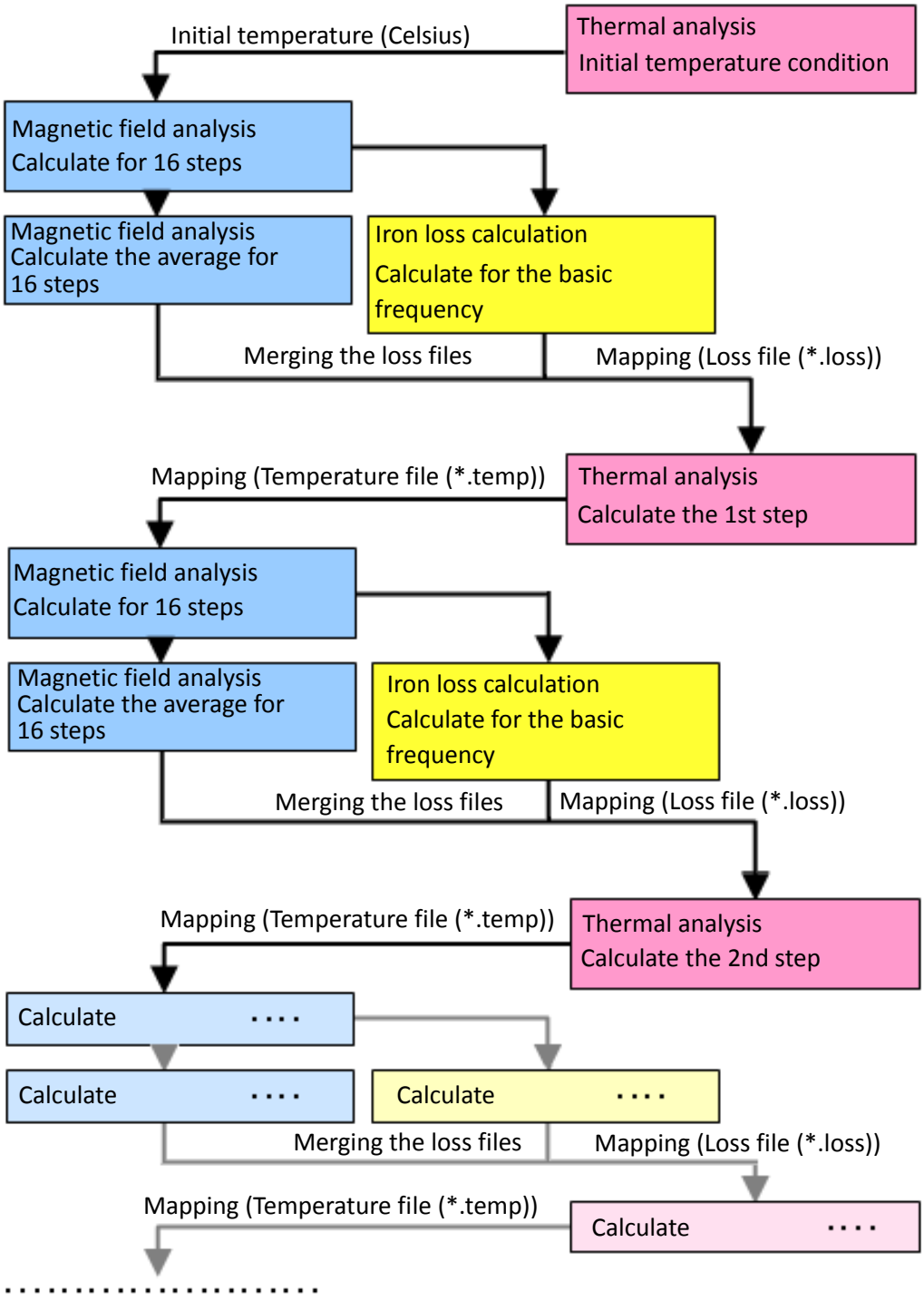
The coupled analysis is run by the following process.

Example 1: If [Loss Averaging] is selected for the coupled analysis, in which a 16-step magnetic field analysis and a 10-step thermal analysis are run.



The average value of the 16-step magnetic field analysis is mapped to the thermal analysis.

Example 2: If [Loss Averaging] is selected and when the iron loss condition is set for magnetic filed analysis at the coupled analysis, in which a 16-step magnetic filed analysis and a 10-step thermal analysis are run.



The sum of the loss calculated from iron loss calculation and the average value of the 16-step magnetic field analysis is mapped to the thermal analysis.

10-5-1-1. Data transferred between solver programs and where it is received

- Magnetic field analysis Joule loss => Thermal analysis, Heat source condition
- Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Magnetic field analysis Joule loss + Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Thermal analysis, Temperature => Magnetic field analysis, Temperature dependent material, Temperature distribution condition

10-5-1-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a two-way coupled analysis is created separately per solver and placed in the same folder.
- The result file for a two-way coupled analysis is exported separately.
  - The results calculated between a thermal analysis and a magnetic field analysis are exported in a series of steps in a result file for a magnetic field analysis (*.jplot, *.plot)
- In the result file (*.jplot, *.plot) of the magnetic field analysis, the results calculated alternately with thermal analysis are output as a series of steps. Time in the magnetic field analysis is output based on the time in the thermal analysis.

  Example: For the case, where: a magnetic field analysis is averaged 0 - 0.1 seconds, a thermal analysis is analyzed every 1 second, 0 - 0.1 seconds, 1 - 1.1 seconds, and 2 - 2.1 seconds are output for the magnetic field analysis.
- This type of coupled analysis can be run from the analysis tool in JMAG-Studio, or from JMAG-Scheduler if using JMAG-Designer.

  ▶▶ Reference
  - For more details on the analysis tool, see "Running the Analysis" in the JMAG-Studio User's Manual.
  - For more information about JMAG-Scheduler, see Chapter 40, "Running the Analysis," in the JMAG-Designer User's Manual.

10-5-1-3. Magnetic Field Analysis (Transient) Settings and Important Notes Regarding the Use of This Function

■ Material

- Creates a material with temperature dependent characteristic that is set to a part.
  - In case temperature dependency of electrical conductivity is put into consideration, set the temperature-electrical conductivity curve.
  - Set a curve for temperature - relative permeability when setting the materials to use the temperature dependency of relative permeability.
  - Set a curve for temperature - magnetic susceptibility when using the temperature dependency for a magnetic property defined by a B-H curve.

▶▶ Reference

For more information, see P.30 "2-3-6. Specifying the Magnetic Properties of a Temperature Dependency" and P.33 "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)".

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Thermal] check box in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Joule Loss Density] check box.
  - Select the [Joule Loss Density] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Joule Loss Density] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.
- It is not necessary to make the number of analysis steps in step conditions coincide with the number of analysis steps in thermal analysis data. Specify the step condition for one cycle of field's time variation for the magnetic field analysis.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency. Select [Two Way] as the coupling type.
- The heat source condition needs to be set when the loss in an FEM coil and resistance component is used for a thermal analysis.
  - If this condition is set for a partial model, the conversion factor needs to be set for both the FEM model and the circuit. The conversion factor can be specified with the analysis control condition in JMAG-Studio and with the study properties in JMAG-Designer.
- The [Basic Frequency] set in the iron loss condition and the step control in magnetic field analysis are independently set. Therefore, even with the iron loss condition, setting of [Basic Frequency], [Rotation Speed], or [Number of Poles] is required to make one cycle of magnetic field fluctuation a subject of calculation.

10-5-1-4. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Model geometry

- When the magnetic field analysis model is axial symmetric, create the thermal analysis model with the Y-axis as the axis of symmetry.

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Magnetic Field/Electromagnetic Wave] radio button under [Analysis Type] in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Temperature (node data)] check box in the thermal analysis control condition for JMAG-Studio.

■ Condition

- Set the heat source condition and select [Two Way] as the coupling type.
  - When a partial model is used, if the heat source type is specified to the total amount for the heat source condition, the loss (total amount of heat source) obtained with magnetic field analysis needs to be converted using one of the following methods.
    - Specify the partial model condition (recommended)
    - Specify the loss scale for the heat source condition
- The motion condition is not available.

10-5-1-5. Settings for Running a Coupled Analysis and Important Notes Regarding the Use of This Function (JMAG-Scheduler, Running Analysis Tool)

■ Use different mesh for each analysis

- When the meshes of the magnetic field and thermal analyses are different, turn on the flag for using different meshes for each analysis.
- A coupled analysis between a magnetic field analysis and thermal analysis can run from a solid model. The setting for using a different mesh between each analysis must also be set.

■ Temperature evaluation section, area losses

- When the magnetic field analysis model is 2D, set the temperature evaluation section and loss generation region. Specify the distance in the Z-axis direction based on the XZ plane for [Position].
- When the magnetic field analysis model is axial symmetric, set the temperature evaluation section. Specify the angle in the periodic direction of the Y center axis based on the XY plane for [Position].

  MEMO For the analysis results of an axial symmetric model, 360-degree mapping is done in the circumferential direction over the thermal analysis 3D model. Therefore, it is not necessary to set the loss generation region.

■ Average loss

- Select the average loss setting.
  The loss for magnetic field analysis is averaged based on the trapezoidal rule when the setting for average loss selected.
- When averaging the loss in the transient response analysis, one period of magnetic field variation is averaged. There are two ways to specify the period.
  - Specify the area to be averaged in step or time in the average loss flag as an option.
  - Specifies the number of steps for a 1-period magnetic field variation (all the steps are averaged in the option to average loss)

  If running a transient analysis using a voltage source, etc., enter a period at the steady state.
- When specifying the range for averaging the loss, enter time (sec) or number of steps from the last step in the transient response analysis.
  - If the number of steps in magnetic field analysis is set to [Specify Displacement], the range cannot be specified by time.
  - If it is set to [External Program], the range cannot be specified.
  - If the time entered does not correspond to any step, JMAG uses the loss value obtained by linearly interpolating the loss between the proximate steps.

■ Start time for applying, end time for applying, number of divisions

- Start time for magnetic field analysis, end time, time interval, and number of steps is specified differently from a thermal analysis.

▶▶ Reference
For more information see, "10-3. Time Interval Settings for Two-way Coupled Analysis (JMAG-Designer)" on page 773.

10-5-2. Magnetic Field Analysis (Transient) <=> Thermal Analysis

This is analysis in the case where magnetic field analysis and thermal analysis are coupled on the same time scale.

In magnetic field analysis, the coupled analysis in both directions is performed through combined use of thermal analysis and transient response magnetic field analysis when an input current (voltage) is arbitrarily applied to a rectangular wave.

10-5-2-1. Data Transferred between Solver Programs and where it is Received

- Magnetic field analysis Joule loss => Thermal analysis, Heat source condition
- Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Magnetic field analysis Joule loss + Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Thermal analysis, Temperature => Magnetic field analysis, Temperature dependent material, Temperature distribution condition

10-5-2-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a two-way coupled analysis is created separately per solver and placed in the same folder.
- The result file for a two-way coupled analysis is exported separately.
- This type of coupled analysis can be run from the analysis tool in JMAG-Studio, or from JMAG-Scheduler if using JMAG-Designer.

▶▶ Reference

- For more details on the analysis tool, see “Running the Analysis” of the JMAG-Studio User's Manual.
- For more information about JMAG-Scheduler, see Chapter 40, “Running the Analysis,” in the JMAG-Designer User's Manual.

10-5-2-3. Magnetic Field Analysis (Transient) Settings and Important Notes Regarding the Use of This Function

■ Material

- Creates a material with temperature dependent characteristic that is set to a part.
  - In case temperature dependency of electrical conductivity is put into consideration, set the temperature-electrical conductivity curve.
  - Set a curve for temperature - relative permeability when setting the materials to use the temperature dependency of relative permeability.
  - Set a curve for temperature - magnetic susceptibility when using the temperature dependency for a magnetic property defined by a B-H curve.

▶▶ Reference

For more information, see P.30 "2-3-6. Specifying the Magnetic Properties of a Temperature Dependency" and P.33 "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)".

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Thermal] check box in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Joule Loss Density] check box.
  - Select the [Joule Loss Density] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Joule Loss Density] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.
- The number of steps and time increment for step control needs to be the same with the magnetic field analysis and thermal analysis.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency. Select [Two Way] as the coupling type.
- The settings for motion region, speed, rotation axis, motion direction, and initial position of rotor need to be the same with the magnetic field analysis and thermal analysis when using the motion condition.

- The heat source condition needs to be set when the loss in an FEM coil and resistance component is used for a thermal analysis.
  - If this condition is set for a partial model, the conversion factor needs to be set for both the FEM model and the circuit. The conversion factor can be specified with the analysis control condition in JMAG-Studio and with the study properties in JMAG-Designer.

10-5-2-4. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Model geometry

- When the magnetic field analysis model is axial symmetric, create the thermal analysis model with the Y-axis as the axis of symmetry.

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Magnetic Field/Electromagnetic Wave] radio button under [Analysis Type] in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Temperature (node data)] check box of the thermal analysis control condition for JMAG-Studio.
- The number of steps and time increment for step control needs to be the same with the magnetic field analysis and thermal analysis.

■ Condition

- The settings for motion region, speed, rotation axis, motion direction, and initial position of rotor need to be the same with the magnetic field analysis and thermal analysis when using the motion condition.
- Set the heat source condition and select [Two Way] as the coupling type.
  - When a partial model is used, if the heat source type is specified for the total amount for the heat source condition, the loss (total amount of heat source) obtained with magnetic field analysis needs to be converted using one of the following methods.
    - Specify the partial model condition (recommended)

- Specify the loss scale for the heat source condition

10-5-2-5. Settings for Running a Coupled Analysis and Important Notes Regarding the Use of This Function (JMAG-Scheduler, Running Analysis Tool)

■ Use different mesh for each analysis

- When the meshes of the magnetic field and thermal analyses are different, turn on the flag for using different meshes for each analysis.
- A coupled analysis between a magnetic field analysis and thermal analysis can run from a solid model. The setting for using a different mesh between each analysis must also be set.

■ Temperature evaluation section, area losses

- If using a 2D model for the magnetic field analysis, specify the temperature evaluation section and loss generated region. Specify the distance in the Z-axis direction based on the XZ plane for [Position].
- If using a axisymmetric model for the magnetic field analysis, specify the temperature evaluation section. Specify the angle in the periodic direction of the Y center axis based on the XY plane for [Position].

MEMO For the analysis results of an axial symmetric model, 360-degree mapping is done in the circumferential direction over the thermal analysis 3D model. Therefore, it is not necessary to set the loss generation region.

10-5-3. Magnetic Field Analysis (Frequency) <=> Thermal Analysis

The time constant of magnetic field analysis is very low as compared with that of thermal analysis, and the coupled analysis in the case where magnetic field analysis can be handled as frequency response analysis is performed.

10-5-3-1. Data transferred between solver programs and where it is received

- Magnetic field analysis Joule loss => Thermal analysis, Heat source condition
- Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Magnetic field analysis Joule loss + Iron loss analysis, Iron loss => Thermal analysis, Heat source condition
- Thermal analysis, Temperature => Magnetic field analysis, Temperature dependent material, Temperature distribution condition

10-5-3-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a two-way coupled analysis is created separately per solver and placed in the same folder.
- The result file for a two-way coupled analysis is exported separately.
- This type of coupled analysis can be run from the analysis tool in JMAG-Studio, or from JMAG-Scheduler if using JMAG-Designer.

▶▶ Reference

- For more information about the analysis tool, see “Running the Analysis” of the JMAG-Studio User's Manual.
- For more information about JMAG-Scheduler, see Chapter 40, “Running the Analysis,” in the JMAG-Designer User's Manual.

MEMO The result from the same frequency which is outputted multiple times can be displayed for confirmation, by setting the X-axis of the graph to [Time].

10-5-3-3. Magnetic Field Analysis (Frequency) Settings and Important Notes Regarding the Use of This Function

■ Material

- Creates a material with temperature dependent characteristic that is set to a part.
  - In case temperature dependency of electrical conductivity is put into consideration, set the temperature-electrical conductivity curve.
  - Set a curve for temperature - relative permeability when setting the materials to use the temperature dependency of relative permeability.
  - Set a curve for temperature - magnetic susceptibility when using the temperature dependency for a magnetic property defined by a B-H curve.

▶▶ Reference
For more information, see P.30 "2-3-6. Specifying the Magnetic Properties of a Temperature Dependency" and P.33 "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)".

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Thermal] check box in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Joule Loss Density] check box.
  - Select the [Joule Loss Density] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Joule Loss Density] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.
- Specify the number of steps in [Frequency Control] to one. If multiple steps are set in a coupled analysis, only the first step is used.
- A magnetic field analysis can be run on multiple frequencies by defining a time dependency of frequency point sequence in a frequency control. The frequency for a magnetic field analysis is determined by the time defined in a step control of a thermal analysis.

▶▶ Reference
For information about time dependency of frequency point sequence, see "3-39. Frequency Control" on page 278.

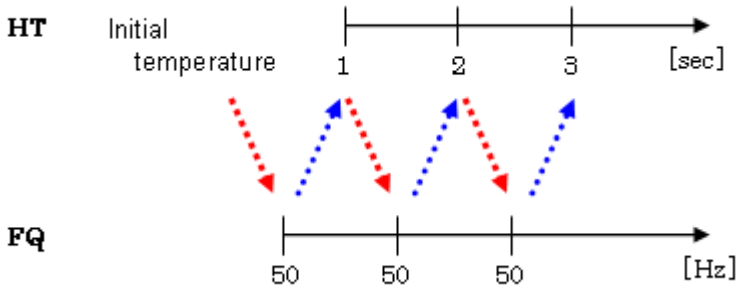
- The electrical power for a magnetic field analysis is determined by the time defined in a step control of a thermal analysis, when a time dependency is specified using a power source component in a circuit.

  ▶▶ Reference

  For more information about the power source component, see "3-16-15. Power Source Component (JMAG-Designer)" on page 172.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency. Select [Two Way] as the coupling type.
- The time information for a thermal analysis is used for the settings when the condition for time dependency is used.
  - The coupled analysis in which temperature and time were considered can be performed by using the power source of time dependency.
  - Specify them by using one of the methods below when using motion conditions.
    - Rotation -> Displacement specification -> Velocity -> Constant
    - Rotation -> Displacement specification -> Velocity -> Point sequence



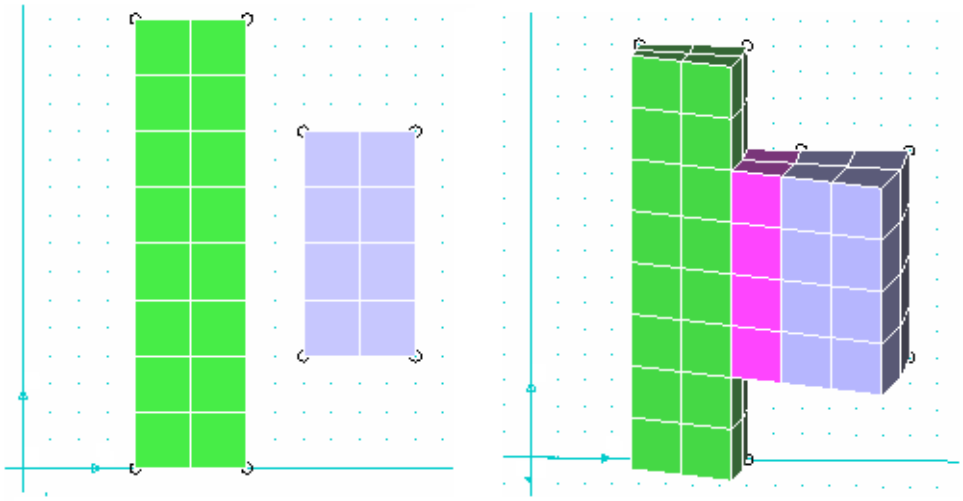
For three-step thermal analysis with 50 Hz specified for magnetic field analysis

- The heat source condition needs to be set when the loss in an FEM coil and resistance component is used for a thermal analysis.
  - If this condition is set for a partial model, the conversion factor needs to be set for both the FEM model and the circuit. The conversion factor can be specified with the analysis control condition in JMAG-Studio and with the study properties in JMAG-Designer.

10-5-3-4. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Model geometry

- When the magnetic field analysis model is axial symmetric, create the thermal analysis model with the Y-axis as the axis of symmetry.



Magnetic field analysis (axial symmetric) Thermal analysis (3D)

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Magnetic Field/Electromagnetic Wave] radio button under [Analysis Type] in the [Analysis Control] dialog box for JMAG-Studio.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Temperature (node data)] check box in the thermal analysis control condition for JMAG-Studio.
- Specify [Steps] in step control. The number of calculation repeated in a thermal analysis and magnetic field analysis coincide the number of steps defined in the thermal analysis.

■ Conditions

- Specify one of the methods below when using motion conditions.
  - Rotation -> Constant

- Rotation -> Point sequence

For the velocity of motion, specify the same value as for magnetic field analysis.

- Set the heat source condition and select [Two Way] as the coupling type.
  - When a partial model is used, if the heat source type is specified for the total amount for the heat source condition, the loss (total amount of heat source) obtained with magnetic field analysis needs to be converted using one of the following methods.
    - Specify the partial model condition (recommended)
    - Specify the loss scale for the heat source condition

10-5-3-5. Settings for Running a Coupled Analysis and Important Notes Regarding the Use of This Function (JMAG-Scheduler, Running Analysis Tool)

■ Use different mesh for each analysis

- When the meshes of the magnetic field and thermal analyses are different, turn on the flag for using different meshes for each analysis.
- A coupled analysis between a magnetic field analysis and thermal analysis can run from a solid model. The setting for using a different mesh between each analysis must also be set.

■ Temperature evaluation section, area losses

- If using a 2D model for the magnetic field analysis, specify the temperature evaluation section and loss generated region. Specify the distance in the Z-axis direction based on the XZ plane for [Position].
- If using a axisymmetric model for the magnetic field analysis, specify the temperature evaluation section. Specify the angle in the periodic direction of the Y center axis based on the XY plane for [Position].

MEMO For the analysis results of an axial symmetric model, 360-degree mapping is done in the circumferential direction over the thermal analysis 3D model. Therefore, it is not necessary to set the loss generation region.

■ Start time for applying, end time for applying, number of divisions

- Start time for magnetic field analysis, end time, time interval, and number of steps is specified differently from a thermal analysis.

▶▶ Reference
For more information see, "10-3. Time Interval Settings for Two-way Coupled Analysis (JMAG-Designer)" on page 773.

10-5-3-6. Procedures for Calculating Multiple Frequencies in the Same Model

Conduct the following procedure when obtaining magnetic field analysis results for both low frequency and high frequency.

1. **Prepare a JCF file for thermal analysis, a JCF file for magnetic field analysis specifying low frequency, and a JCF file for magnetic field analysis specifying high frequency.**
2. **Start JMAG-Scheduler.**
3. **Click [Job] -> [Add Job].**

   The [Add Job] dialog box is displayed.
4. **Select the JCF file for the thermal analysis in step 1, and then click [OK].**

   The [Submit to Queue] dialog box is displayed.
5. **Click [Add File].**

   The [Coupled Files] dialog box is displayed.
6. **Click [Browse], and then select the JCF file for a magnetic field analysis with low frequency that was specified in step 1.**

   Using the same procedures, specify the JCF file for a magnetic field analysis with high frequency that was specified in step 1.
7. **Specify the coupled analysis settings in each JCF file for a magnetic field analysis as required.**

   For information about settings, see "10-5-3-5. Settings for Running a Coupled Analysis and Important Notes Regarding the Use of This Function (JMAG-Scheduler, Running Analysis Tool)" on page 799, and "10-3. Time Interval Settings for Two-way Coupled Analysis (JMAG-Designer)" on page 773.
8. **Click [OK].**

   The [Coupled Files] dialog box closes and the [Submit to Queue] dialog box is displayed.
9. **Click the [Submit Job] button.**

   The calculation for the added job is automatically run.

10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction

■ Necessary solvers

- Magnetic field analysis (one of the following)
  - Static magnetic field analysis (ST)
  - Frequency response magnetic field analysis (FQ)
  - Transient response magnetic field analysis (TR, DP)
  - Iron loss analysis (LS)
    *Only in the case of static structural analysis => iron loss analysis
- Structural analysis (DS) (one of the following)
  - Static structural analysis
  - Frequency response structural analysis

■ Types of analysis

- Static magnetic field analysis => Static structural analysis
- Transient response magnetic field analysis => Static structural analysis, Frequency response structural analysis
- Frequency response magnetic field analysis => Frequency response structural analysis, Frequency response structural analysis
- Static structural analysis => Magnetic field analysis *Including the Abaqus link

10-6-1. Magnetic Field Analysis (Static) => Structural Analysis (Static), Magnetic Field Analysis (Transient) => Structural Analysis (Static/Frequency)

The load for a structural analysis is determined, by using an electromagnetic force for each node calculated from a magnetic field analysis. Magnetostrictive force calculated from a magnetic field analysis can also be used.

10-6-1-1. Data transferred between solver programs and where it is received

- Magnetic field analysis, electromagnetic force => Structural analysis, force condition
- Magnetic field analysis, magnetostrictive force => Structural analysis, force condition

10-6-1-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a coupled analysis is created separately per solver.
- The nodal force in a specified section is the electromagnetic force exported from a magnetic field analysis (transient response analysis). The electromagnetic force is resolved into amplitude and phase for each frequency by the Fast Fourier Transform and is used for a structural analysis.
- Magnetostrictive force outputted from a magnetic field analysis can be used.

  ▶▶ Reference
  For more information about calculating magnetostrictive force in a magnetic field analysis, see "3-23. Magnetostriction Calculation (JMAG-Designer)" on page 232.
- The value interpolated by the electromagnetic force from another frequency is used for the structural analysis when the electromagnetic force for the frequency component that is specified in the frequency control (frequency list condition for JMAG-Studio) does not exist for the structural analysis.

10-6-1-3. Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

■ Material

- The magnetostriction properties are set when using magnetostrictive force in a structural analysis.

■ Model geometry

- The result file of magnetic field analysis using an axisymmetric model cannot be referenced by the structural analysis. A 2D model or 3D model needs to be created.

■ Study properties / Analysis control

- Select the [Nodal Force] check box.
  - Select the [Nodal Force] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Nodal Force] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.

■ Conditions

- If the motion condition is set for the magnetic field analysis, the electromagnetic force cannot be mapped for the model of structural analysis. For example, if the mover of a motor is used for the structural analysis, the motion of the mover needs to be simulated relatively by setting the motion condition for the stator in the magnetic field analysis.
- Sets the condition to calculate electromagnetic force/torque, when using electromagnetic force in a structural analysis.
  Set the condition to calculate electromagnetic force or torque that is used to calculate the electromagnetic force for the structural analysis if necessary.
- Specifies settings for the magnetostriction condition, when using magnetostrictive force in a structural analysis.

10-6-1-4. Structural Analysis Settings and Important Notes Regarding the Use of This Function

■ Condition

- The result file of magnetic field analysis is specified for the electromagnetic force condition.
  - For JMAG-Designer, specify a jplot or plot file.
  - For JMAG-Studio, specify a plot file.

  MEMO Specify settings for multiple force conditions when referencing both electromagnetic force and magnetostrictive force in a structural analysis.
- To reference a plot file, the input file is required.
  - For JMAG-Designer, specify a JCF file.
  - For JMAG-Studio, specify the JCF file or solver data file.

- If the adaptive mesh or patch mesh function is used in the magnetic field analysis, specify the "*_input.jcf" file that is exported to the same folder as the input file after the magnetic field analysis is complete. This file contains the mesh data in addition to the input file.

  MEMO The asterisk "*" in the "*_input.jcf" indicates the name of the JCF file that is used for running the magnetic field analysis.

- For JMAG-Studio, the solver data file of structural analysis to which the electromagnetic force condition is set cannot be used to run the analysis using Analysis Tool. In this case, the analysis can be run from the graphics window with the project file of structural analysis open.
  To run the analysis using Analysis Tool, use the JCF file instead of the solver data file.

■ The geometry of the mesh is different in a magnetic field analysis and structural analysis (JMAG-Designer)

- Electromagnetic force cannot be transferred from the solid element model of magnetic field analysis to the shell element model of structural analysis.
- For JMAG-Designer, electromagnetic force can be transferred between parts of magnetic field model and structural model. The outline of parts needs to be the same between the magnetic field analysis and the structural analysis.
- JMAG-Designer automatically detects differences in mesh geometry where special settings are not required.

■ The geometry of the mesh is different in a magnetic field analysis and structural analysis (JMAG-Studio)

- Electromagnetic force cannot be transferred from the solid element model of magnetic field analysis to the shell element model of structural analysis.
- The areas of electromagnetic force should be the same between the magnetic field analysis and the structural analysis.
- For JMAG-Studio, the [Use different mesh between magnetic field and structure] check box needs to be selected for the electromagnetic force condition.
- For JMAG-Studio, the material IDs of electromagnetic force should be the same between the magnetic field analysis and the structural analysis.
- For JMAG-Studio, the areas of electromagnetic force need to be continuous. Electromagnetic force cannot be transferred to several independent areas.

■ Referring to a result file for a magnetic field analysis in a partial model

- If the magnetic field analysis uses a partial model and the structural analysis uses a full model or a partial model larger than the partial model of magnetic field analysis, the analysis results of magnetic field analysis need to be converted to values of full model.
  - For JMAG-Designer, a partial model is automatically converted to a full model based on the periodic boundary condition of magnetic field analysis. Therefore, no setting is required. The rotation axis required to convert a partial model to a full model is the Z-axis in the global coordinate system.
  - In JMAG-Studio, the [Full Model Extension] check box needs to be selected for the electromagnetic force condition. The rotation axis required to extend a partial model to a full model is the Z-axis in the global coordinate system.

  However, this function cannot be used for the following magnetic field analysis models.
  - An axisymmetric model is not supported.
  - A model with a translation periodicity is not supported. A model with a rotation periodicity is supported.

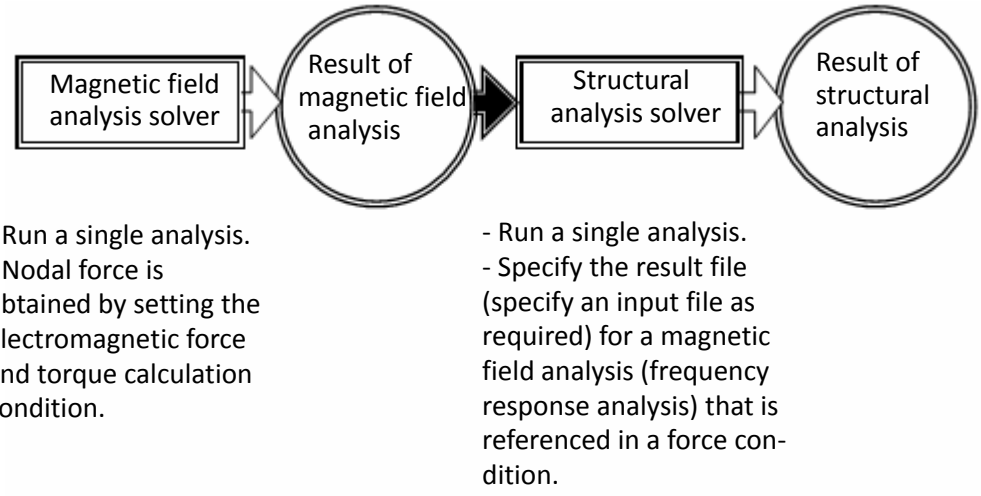
■ Referring to a result file for a 2D magnetic field analysis

- If a 2D model is used for the magnetic field analysis, the analysis results are exported for 1 m of thickness. Therefore, the magnetic field analysis results need to be converted if the model for both the magnetic field analysis and structural analysis are 2D.
  - For JMAG-Designer, the electromagnetic force is automatically converted based on [Thickness] set in the shell properties of each part (shell).
    However, the electromagnetic force is not converted if [Thickness] does not match for multiple parts (shells) the force condition is set.
  - For JMAG-Studio, the thickness should be scaled to the thickness specified in the Shell Element Property dialog box.
    For example, if [Thickness] is specified to 1 mm for the shell element property condition, enter “0.001” in the [Scale Factor] box for the electromagnetic force condition.
- The mapping region for the electromagnetic force for the structural analysis model needs to start on the XY plane along the +Z-axis direction if the magnetic field analysis results to refer are for a 2D model and the structural analysis model is a 3D model.
- If a 2D model is used for the magnetic field analysis and a 3D model is used for the structural analysis, the results of magnetic field analysis need to be scaled and mapped to the model of structural analysis.

- For JMAG-Designer, the coordinate system referenced by the electromagnetic force condition needs to be selected. The results of magnetic field analysis are mapped in the Z-axis direction of the coordinate system.
- For JMAG-Studio, the [Extension from 2D (Magnetic Field) to 3D (Structure)] check box needs to be selected, and the height of the structural analysis model needs to be specified in the [Extend Distance] box. The direction of extension is the Z-axis direction in the global coordinate system.

10-6-2. Magnetic Field Analysis (Frequency) -> Structural Analysis (Static/ Frequency)

Nodal force calculated from a magnetic field analysis (frequency response analysis) can be used as input load for a structural analysis (static/frequency response analysis). Magnetostrictive force calculated from a magnetic field analysis can also be used.


Magnetic field analysis solver
Result of magnetic field analysis
Structural analysis solver
Result of structural analysis
- Run a single analysis.
- Nodal force is obtained by setting the electromagnetic force and torque calculation condition.
- Run a single analysis.
- Specify the result file (specify an input file as required) for a magnetic field analysis (frequency response analysis) that is referenced in a force condition.

Concept of the coupled magnetic field and structural analyses

10-6-2-1. Data transferred between solver programs and where it is received

- Magnetic field analysis, electromagnetic force => Structural analysis, force condition
- Magnetic field analysis, Lorentz force density => Structural analysis, force condition
- Magnetic field analysis, magnetostrictive force => Structural analysis, force condition

10-6-2-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a coupled analysis is created separately per solver.
- The electromagnetic force for the magnetic field analysis is outputted by an effective value, 0-order component, and 2-order component (electromagnetic force cannot be outputted by amplitude and phase). These values are used in a structural analysis. The phase of the electromagnetic force in the magnetic field analysis is not used in the structural analysis in which the phase is calculated as zero.
- The Lorentz force density and the electromagnetic force outputted from a magnetic field analysis are used.
  - The value for a 0-order component in an electromagnetic force/Lorentz force density is used in a structural analysis (static analysis).

  - The value for a 0-order component and 2-order component in an electromagnetic force/Lorentz force density is used in a structural analysis (frequency response analysis).
- Magnetostrictive force outputted from a magnetic field analysis can be used.

  ▶▶ Reference

  For more information about calculating magnetostrictive force in a magnetic field analysis, see "3-23. Magnetostriction Calculation (JMAG-Designer)" on page 232.
- The value interpolated by the electromagnetic force from another frequency is used for the structural analysis when the electromagnetic force for the frequency component that is specified in the frequency control (frequency list condition for JMAG-Studio) does not exist for the structural analysis.

10-6-2-3. Magnetic Field Analysis (Frequency) Settings and Important Notes Regarding the Use of This Function

■ Material

- The magnetostriction properties are set when using magnetostrictive force in a structural analysis.

■ Study properties / Analysis control

- Select the [Nodal Force] check box.
  - Select the [Nodal Force] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Nodal Force] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.
- Select the output flag for the Lorentz force density.
  - Select the [Lorentz Force Density] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Lorentz Force Density] check box in the [Control Output Items] tab of the magnetic field analysis control condition for JMAG-Studio.

■ Condition

- Sets the condition to calculate electromagnetic force/torque, when using electromagnetic force in a structural analysis.
  Set the condition to calculate electromagnetic force or torque that is used to calculate the electromagnetic force for the structural analysis if necessary.

- Specifies settings for the magnetostriction condition, when using magnetostrictive force in a structural analysis.

10-6-2-4. Structural Analysis (Static Analysis/Frequency Response Analysis) Settings and Important Notes Regarding the Use of This Function

■ Condition

- Specify the result file of frequency response analysis for the electromagnetic force condition.
  - For JMAG-Designer, specify a plot file or Jplot file.
  - For JMAG-Studio, specify a plot file.

  MEMO Specify settings for multiple force conditions when referencing both electromagnetic force and magnetostrictive force in a structural analysis.
- To reference a plot file, specify the input file of frequency response magnetic field analysis.
  - For JMAG-Designer, specify a JCF file.
  - For JMAG-Studio, specify a JCF file or solver data file.
  - If the adaptive mesh or patch mesh function is used in the magnetic field analysis, specify the "*_input.jcf" file that is exported to the same folder as the input file after the magnetic field analysis is complete. This file contains the mesh data in addition to the input file.

    MEMO The asterisk "*" in the "*_input.jcf" indicates the name of the JCF file that is used for running the magnetic field analysis.

■ Referring to a result file for a 2D magnetic field analysis

- The mapping region for the electromagnetic force for the structural analysis model needs to start on the XY plane along the +Z-axis direction if the magnetic field analysis results to refer are for a 2D model and the structural analysis model is a 3D model.

10-6-3. Structural Analysis => Magnetic Filed Analysis, Iron Loss Analysis

The stress values obtained from a structural analysis can be used for a magnetic field analysis (iron loss analysis).

10-6-3-1. Data transferred between solver programs and where it is received

- Structural analysis Stress => Magnetic field analysis, Iron loss analysis Stress dependency material, Stress dependent material
- Structural analysis Displacement => Magnetic field analysis Displacement condition

10-6-3-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a coupled analysis is created separately per solver.
- Depending on the model dimensions that are used the magnetic field analysis (iron loss analysis) and structural analysis cannot be coupled.

		Magnetic field analysis, iron loss analysis		
		2D model	3D model	Ax model
Structural analysis	2D model	O	X	X
	3D model	O[*1]	O	X

O: Model dimensions that can be coupled

X: Model dimensions that cannot be coupled.

[*1]: The stress distribution refers to the value of the same position as the elements for magnetic field analysis (iron loss analysis).

- The same mesh model is recommended to use for both magnetic field analysis (iron loss analysis) and structural analysis.
  Even if a different mesh model is used between magnetic field analysis (iron loss analysis) and structural analysis, the calculation can be performed. In this case, the model geometry of the structural analysis needs to include the model geometry of the magnetic field analysis.
  Examples)
  - If the structural analysis to be referenced uses a 1/2 model and the magnetic field analysis (iron loss analysis) uses a 1/4 model = = = = = > Can be analyzed.
  - If the structural analysis to be referenced uses a full model and the magnetic field analysis (iron loss analysis) uses a 1/2 model = = = = = > Can be analyzed.

- If the structural analysis to be referenced uses a 1/4 model and the magnetic field analysis (iron loss analysis) uses a 1/2 model = = = = = > Cannot be analyzed.

Although the calculation can be performed when the different mesh model is used for each analysis, the calculation accuracy may be reduced compared with when the same mesh model is used for both analyses. When evaluating the results, you need to take this into consideration.

- The stress-dependent magnetic properties and iron loss curves refer to principle stress III.

10-6-3-3. Structural Analysis Settings and Important Notes Regarding the Use of This Function

■ Study properties / Analysis control

- Specify [Stress] when referring to the stress in a magnetic field analysis.
  - Select the [Stress] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
- Specify [Displacement] when referring to the displacement in magnetic field analysis.
  - Select the [Displacement] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.

10-6-3-4. Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

■ Material

- Creates a material with stress dependent characteristics that is set to a part when using stress.
  - The stress value obtained from a structural analysis is ignored when materials with stress dependent characteristics are not used.

▶▶ Reference
For more information, see P.29 "2-3-5. Specifying the Magnetic Properties of a Stress Dependency".

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.

■ Condition

- For the stress distribution condition of magnetic field analysis (iron loss analysis), select the [Use Structural Analysis Result File] radio button, and specify the result file of a structural analysis.
  - Specifies the JPLOT file, PLOT file, or ODB file (Abaqus result file) in JMAG-Designer.
    Specify the unit of stress, length, and the Abaqus step when specifying an ODB file (Abaqus result file (*.jplot)).

    ▶▶ Reference
    - For more information about the stress distribution condition, see P.253 "3-28. Stress Distribution".
    - For more information about running a coupled analysis in Abaqus, see "Chapter 11. Coupled Analysis Using Abaqus" on page 823.
  - A plot file can be selected in JMAG-Studio.
- Set the displacement condition when using displacement. Select [One Way] as the coupling type and specify the result file to be referenced.
- The stress distribution condition and the displacement condition can be used together. This setting can be used for each conditions referring to a different structural analysis file.

10-7. Coupling Thermal Analysis and Electric Field Analysis in One Direction

■ Required solver

- Thermal analysis (HT)
- Electric field analysis (EL) (one of the following)
  - Current distribution analysis
  - Electric field analysis (frequency response analysis)

■ Types of analyses

- Current distribution analysis => Thermal analysis
- Electric field analysis (frequency response analysis) => Thermal analysis
- Thermal analysis => Current distribution analysis

10-7-1. Current Distribution Analysis, Electric Field Analysis (Frequency) => Thermal Analysis

Calculate the power consumption density for an electric field analysis (frequency response analysis) and a current distribution analysis to run an independent steady state or a transient state analysis.

10-7-1-1. Data transferred between solver programs and where it is received

- Current distribution analysis Power Consumption Density => Thermal Analysis Heat source condition
- Electric field analysis (frequency response analysis) Power consumption density => Thermal analysis Heat source condition

10-7-1-2. Important Notes Regarding the Use of This Function

- The temperature dependency of an electric conductivity cannot be used.
- The input file (*.jcf) for a coupled analysis is created separately per solver.
- The result file for a two-way coupled analysis is exported separately.
- The geometry of a mesh used for a thermal analysis and electric field analysis needs to be the same.

10-7-1-3. Electric Field Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select [Normal Run] under [Analysis Type] in the electric field analysis control condition for JMAG-Studio.
- Select the [Power Consumption] check box.
  - Select the [Power Consumption] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Power Consumption (element data)] check box in the electric field analysis control condition for JMAG-Studio.

10-7-1-4. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select [Normal Run] under [Analysis Type] in the thermal analysis control condition for JMAG-Studio.

■ Condition

- Set the heat source condition and select [One Way] as the coupling type and [Distribution] as the heat source type. Specify a result file of a electric field analysis and a loss file (*.loss) exported as a result for a electric field analysis.

  MEMO The geometry of an input data and loss file needs to be the same when specifying a loss file. A map is not required in which a JCF file or a solver data file does not need to be specified.

  MEMO If a plot file is selected for [Result File (jplot/plot/loss files)], a JCF file needs to be specified.

10-7-2. Thermal Analysis => Current Distribution Analysis

Run a current distribution analysis using the temperature dependent electric conductivity after calculating the temperature in a thermal analysis.

10-7-2-1. Data transferred between solver programs and where it is received

Thermal analysis, Temperature => Current distribution analysis, Temperature dependent material, Temperature distribution condition

10-7-2-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a coupled analysis is created separately per solver.
- The geometry of a mesh used for a thermal analysis and electric field analysis needs to be the same.

10-7-2-3. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for a study property.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Temperature (node data)] check box of the thermal analysis control condition for JMAG-Studio.

10-7-2-4. Current Distribution Analysis and Notes Regarding the Use of this Function

■ Material

- Creates a material with temperature dependent characteristic that is set to a part.
  - Set a temperature - electric conductivity curve as an electric property for material settings when using temperature dependency for electric conductivity.

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency. Select [One Way] as the coupling type and specify the result file (*.jplot, *.plot) for a thermal to be referenced.

10-8. Coupling Thermal Analysis and Electric Field Analysis in Both Directions

■ Necessary solvers

- Thermal analysis (HT)
- Electric field analysis (EL) (one of the following)
  - Current distribution analysis
  - Electric field analysis (frequency response analysis)

■ Types of analysis

- Thermal analysis <=> Current distribution analysis
- Thermal analysis <=> Electric field analysis (frequency response analysis) * Supported only for JMAG-Studio

10-8-1. Thermal Analysis <=> Current Distribution Analysis, Electric Field Analysis (Frequency)

Run a two-way coupled analysis by combining HT and EL when there is electric conductivity with temperature dependency in a current distribution analysis and an electric field analysis (frequency response analysis).

MEMO Thermal analysis <=> Electric field analysis (frequency response analysis) is only supported by JMAG-Studio.

10-8-1-1. Data transferred between solver programs and where it is received

- Thermal analysis, Temperature => Current distribution analysis, Temperature dependent material, Temperature distribution condition
- Current distribution analysis Power Consumption Density => Thermal Analysis Heat source condition
- Electric field analysis (frequency response analysis) Power consumption density => Thermal analysis Heat source condition

10-8-1-2. Important Notes Regarding the Use of This Function

- The input file (*.jcf) for a two-way coupled analysis is created separately per solver and placed in the same folder.
- The result files (JPLOT file or PLOT file) for the thermal analysis and the current distribution analysis are exported in different files.

- For JMAG-Studio, this coupled analysis is run from the Analysis Tool. For JMAG-Designer, this coupled analysis is run from JMAG-Scheduler.

▶▶ Reference

- For more information about the Analysis Tool, see Chapter 6, "Running the Calculation," in the JMAG-Studio User's Manual.
- For more information about JMAG-Scheduler, see Chapter 40, "Running the Analysis," in the JMAG-Designer User's Manual.

10-8-1-3. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select [Coupling with Magnetic Field / Electromagnetic Wave] under [Analysis Type] in the thermal analysis control condition for JMAG-Studio.
- Specify [Steps] in step control. The time for the transient state analysis is determined by the number of steps set in the thermal analysis data.
  - Specify the [Steps] text box in the [Step Control] screen of [Study Properties] for JMAG-Designer.
  - Specify the [Steps] text box in a thermal analysis control condition for JMAG-Studio.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.
  - Select the [Temperature (node data)] check box in the thermal analysis control condition for JMAG-Studio.

■ Condition

- Set the heat source condition and select [Two Way] as the coupling type.

10-8-1-4. Current Distribution Analysis, Electric Field Analysis (Frequency) Settings

■ Material

- Creates a material with temperature dependent characteristic that is set to a part.
  - Specify the temperature-electric conductivity (resistivity) curve in the material settings to account for temperature dependent electric conductivity (resistivity) in the current distribution analysis.
    - For JMAG-Designer, select [Electric Conductivity] or [Resistivity] from the [Conductivity Type] list on the [Electric Properties] tab in the [Material Editor] dialog box. In addition, select the [Temperature Dependent] radio button, and then set the temperature-electric conductivity (resistivity) curve.
    - For JMAG-Studio, select the [Nonlinear (E-epsilon table)] radio button under [Relative Permittivity/Electric Conductivity] in the [Material] dialog box, and then set the temperature-electric conductivity curve.

  MEMO For the electric field analysis (frequency response analysis), the temperature dependency of the electric conductivity (resistivity) cannot be taken into account.

■ Study properties / Analysis control

- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - Select the [Coupling with Thermal] radio button under [Analysis Type] in the [Analysis Control] dialog box for JMAG-Studio.
- Set [Steps] to '1'.
  - For JMAG-Studio, enter “1” in the [Steps] text box in the [Analysis Control] dialog box.
  - For JMAG-Designer, enter “1” in the [Steps] under [Step Control] in the [Study Properties] dialog box.

▶▶ Reference

For the setting items of the [Execute] dialog box, refer to the JMAG-Studio User's Manual, “Chapter 6 Run the Calculation”. For JMAG-Designer, see Chapter 40, “Running Analysis,” in the JMAG-Designer User's Manual.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency and select [Two Way] as the coupling type.

10-8-1-5. Settings for Running a Coupled Analysis and Important Notes Regarding the Use of This Function (JMAG-Scheduler, Running Analysis Tool)

■ Use different mesh for each analysis

- Select [Use Different Mesh] when a thermal analysis and a current distribution analysis have different mesh geometry.
- A coupled analysis between a thermal analysis and current distribution analysis can run from a solid model. The setting for using a different mesh between each analysis must also be set.

10-9. Coupling Thermal Analysis and Structural Analysis in One Direction

■ Necessary solvers

- Thermal analysis (HT)
- Structural analysis (DS)
- Tool: Thermal stress analysis converter * JMAG-Studio Only

■ Types of analyses

- Thermal analysis =>Structural analysis

Important **The temperature for structural analysis is used in JMAG-Studio.**

▶▶ Reference

For more information about settings and important notes on temperature for structural analysis, see P.952 "12-8. Temperature for Structural Analysis (JMAG-Studio)".

10-9-1. Thermal Analysis => Structural Analysis (JMAG-Designer)

Import a temperature data from a thermal analysis result into a structural analysis and run a thermal stress analysis.

10-9-1-1. Data transferred between solver programs and where it is received

- Thermal analysis temperature => Structural analysis Temperature load condition

10-9-1-2. Thermal Analysis Settings and Notes Regarding the Use of this Function

■ Study properties / Analysis control

- Clear the two-way coupled analysis.
  - Clear the [Two way coupled analysis] check box in the [Coupled Analysis] screen for a study property.
- Select the [Temperature] check box.
  - Select the [Temperature] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.

10-9-1-3. Structural Analysis Settings and Important Notes Regarding the Use of This Function

■ Condition

- Specifies a temperature load condition. Select [One Way] as the coupling type and specify the result file (*.jplot, *.plot) for a thermal to be referenced.
  - Run a thermal analysis with multiple steps when selecting a temperature difference distribution as a method for temperature.

Chapter 11. Coupled Analysis Using Abaqus

JMAG is able to run a coupled analysis with the unified finite element analysis software Abaqus.

This chapter describes the important notes when running a coupled analysis using JMAG and Abaqus.

11-1. Types of Coupled Analysis

The three main types of coupled analysis for JMAG and Abaqus are the following.

- **One-way coupled analysis (JMAG > Abaqus)**

  Runs an analysis using Abaqus with results by JMAG (loss density, nodal force).

- **One-way coupled analysis (Abaqus-> JMAG)**

  Runs an analysis using JMAG with results by Abaqus (stress).

- **Two-way coupled analysis (JMAG <-> Abaqus)**

  JMAG and Abaqus runs an analysis simultaneously while transferring data from one another.

11-2. One-way Coupled Analysis (JMAG > Abaqus)

Runs an analysis using Abaqus with results by JMAG (loss density, nodal force).

■ Types of analysis

- JMAG magnetic field analysis > Abaqus structural analysis
- JMAG magnetic field analysis, iron loss analysis > Abaqus thermal analysis

■ Required solver (JMAG magnetic field analysis > Abaqus structural analysis)

- JMAG
  - Magnetic field analysis (one of the following)
    - Static analysis (ST)
    - Transient response analysis (TR, DP)
    - Frequency response analysis (FQ)
  - Force mapping tool for Abaqus
- Abaqus
  - Structural analysis
    - Static (static analysis)
    - Steady State Dynamics (frequency dependent)
    - Modal Dynamic (time dependent)

■ Required solver (JMAG magnetic field analysis, iron loss analysis > Abaqus thermal analysis)

- JMAG
  - Magnetic field analysis (one of the following)
    - Static analysis (ST)
    - Transient response analysis (TR, DP)
    - Frequency response analysis (FQ)
    - Iron loss calculation (LS)
  - Heat Generation Mapping Tool for Abaqus
- Abaqus
  - Heat Transfer Analysis
    - Heat Transfer

11-2-1. JMAG Magnetic Field Analysis (Force) > Abaqus Structural Analysis

The electromagnetic force and magnetostrictive force calculated from a magnetic field analysis (static analysis, transient response analysis, frequency response analysis) from JMAG is used in a structural analysis in Abaqus.

11-2-1-1. Flow of Analysis

The following is the flow of analysis.

1. **Calculate an electromagnetic force and magnetostrictive force by running a magnetic field analysis in JMAG.**
2. **Prepare an input file for Abaqus.**
3. **Map the electromagnetic force in step 1 to the input file from Abaqus in step 2. by using the force mapping tool for Abaqus in JMAG.**
4. **Run a structural analysis for Abaqus by using the file in step 3.**

11-2-1-2. Data transferred between programs and where it is received

- JMAG magnetic field analysis, electromagnetic force => Abaqus structural analysis
- JMAG magnetic field analysis, Lorentz force density => Abaqus structural analysis
- JMAG magnetic field analysis, magnetostrictive force => Abaqus structural analysis

11-2-1-3. Important Notes Regarding the Use of This Function

- Mapping of the force distribution is supported.
  - The force is mapped using the of reference interval of electromagnetic force specified in the magnetic field analysis results.
  - Fast Fourier transform (FTT) is used if the Abaqus input file (*.inp) to map transient response magnetic field analysis results is Steady State Dynamics.
  - The electromagnetic force and Lorentz force density is used when the result of JMAG is referred to a magnetic field analysis (frequency response analysis).
    - The 0-order component for an electromagnetic force/Lorentz force is used when ‘Static’ is specified.
    - The 0-order component and 2-order component in an electromagnetic force/Lorentz force density is used when ‘Steady State Dynamics’ and ‘Modal Dynamic’ is specified.
- Models in JMAG and Abaqus that have a different dimensions or divisions are supported.

11-2-1-4. JMAG Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

- This function is not supported for an axisymmetric model in which a 2D model or a 3D model needs to be created.
- A partial model is expanded to full model based on the settings in the periodic boundary condition. The rotation axis when expanding is the Z-axis in the global coordinate system.
  However, this function cannot be used with the following models.
  - Partial model with axisymmetric
  - Partial model with translation periodicity
- To calculate electromagnetic force required for structural analysis, the electromagnetic force and torque calculation condition needs to be set for the magnetic field analysis. The electromagnetic force for the magnetic field analysis is outputted by an effective value, 0-order component, and 2-order component (electromagnetic force cannot be outputted by amplitude and phase).
- Outputting Lorentz force density is optional.
- If the motion condition is set for the magnetic field analysis, the electromagnetic force cannot be mapped for the model of structural analysis.
  For example, if the mover of a motor is used for the structural analysis, the motion of the mover needs to be simulated relatively by setting the motion condition for the stator in the magnetic field analysis.

11-2-1-5. Running Procedures for the Force Mapping Tool for Abaqus

1. **Select [Tools] > [Force Mapping Tool for Abaqus] from the menu bar of JMAG-Designer.**

   The [Force Mapping Tool for Abaqus] dialog box is displayed.

2. **Click [Browse] next to [Structural Input File], and then select the Abaqus file (*.inp) for mapping the electromagnetic force.**

3. **Click [Browse] next to [Output File], and then enter the name and folder to save the Abaqus file (*.inp) after mapping.**

4. **Set the units (length, force, force density) in [Abaqus Units].**

5. Specify the structural analysis data specified in step 2 in [Structural Analysis Model].

- [Target]: Displays a list of the sets specified in the Abaqus input file. Select the set to map the electromagnetic force.

  All of the sets in the list can be selected or cleared using the [Selection Of All/Non-Selection of All] check box.
- [Scaling Factor]: Specifies the scaling factor for the electromagnetic force to map. The value multiplied by the scaling factor is mapped.

  Specify the JMAG magnetic field analysis data to refer in [Magnetic Field Analysis Model].

6. Specify a value in the [Basic Frequency] box in [Magnetic Field Analysis Model], when specifying a frequency response analysis file in [Structural Analysis Model].

The frequency for a magnetic field analysis result needs to be multiply the value of a basic frequency.

7. Click the [Add] button to display the [Edit] dialog box in [Magnetic Field Analysis Model], and then specify and set related settings for a result file of a magnetic field analysis.

Specify the following items in the [Edit] dialog box.

- [Reference Interval]: Specifies the steps in the results to use in the structural analysis from the results file of the magnetic field analysis to refer.

  If the magnetic field analysis results to refer are for a static analysis, the first step for the electromagnetic force is used automatically.

  If the magnetic field analysis results to refer are for a transient response analysis or frequency response analysis, enter the step number in the [Start Step] and [End Step] text box.
- [FFT Resolution]: Specifies the time resolution of the fast Fourier transform If the Abaqus structural analysis is a frequency response analysis and the JMAG magnetic field analysis to refer is a transient response analysis.

  If a value larger than 1 is specified, the time resolution for FFT increases.
- [Extended Distance]: Maps to the structural analysis model by extending the magnetic field analysis results in one direction if the JMAG magnetic field analysis model is 2D and the Abaqus structural analysis model is 3D. The results of magnetic field analysis are mapped in the Z-axis direction of the coordinate system.

8. Specify each item in the [Edit] dialog box, and then click [OK].

The [Force Mapping Tool for Abaqus] dialog box is displayed. The result file in a magnetic field analysis that is referenced to a [Magnetic Field Analysis Model] table is added.

9. Repeat steps 7 and 8 to specify multiple result files for a magnetic field analysis as required.

- Select a file from the [Magnetic Field Analysis Model] table in the [Force Mapping Tool for Abaqus] dialog box, and then click [Edit] when modifying the settings in a result file for an added magnetic field analysis.
- Select a file from the [Magnetic Field Analysis Model] table, and then click [Delete] when deleting an result file of a magnetic field analysis.

10.Click [Run].

The mapping starts.

11-2-1-6. Abaqus Structural Analysis Settings and Important Notes Regarding the Use of This Function

- An analysis step from either 'Static,' 'Steady State Dynamics,' or 'Modal Dynamic' needs to be defined in an Abaqus input file for a map. An electromagnetic force is mapped for the analysis step that is first set in an Abaqus input file for 'Static,' 'Steady State Dynamics,' and 'Modal Dynamic.'
- The dimensions of the regions to map the electromagnetic force need to match in the magnetic field analysis model and the Abaqus structural analysis model.
- The regions to map the electromagnetic force need to be set. Assign the sections for the set of regions (element set) in the Abaqus structural analysis. [Material ID] displays the number identifying the material. The electromagnetic force can be mapped for sets displayed in [Material ID].

■ Element types that can be used in Abaqus

The following are element types that can be used in Abaqus when it is linked with JMAG.

Type of dimension	Type of element	Description
3D	C3D4	Stress/displacement elements(4-node linear tetrahedron)
	C3D6	Stress/displacement elements(6-node linear triangular prism)
	C3D8	Stress/displacement elements(8-node linear brick)
	C3D10	Stress/displacement elements(10-node quadratic tetrahedron)
	C3D15	Stress/displacement elements(15-node quadratic triangular prism)
	C3D20	Stress/displacement elements(20-node quadratic brick)
	DC3D4	Diffusive heat transfer or mass diffusion elements(4-node linear tetrahedron)
	DC3D6	Diffusive heat transfer or mass diffusion elements(6-node linear triangular prism)
	DC3D8	Diffusive heat transfer or mass diffusion elements(8-node linear brick)
	DC3D10	Diffusive heat transfer or mass diffusion elements(10-node quadratic tetrahedron)
	DC3D15	Diffusive heat transfer or mass diffusion elements(15-node quadratic triangular prism)
	DC3D20	Diffusive heat transfer or mass diffusion elements(20-node quadratic brick)
	DCC3D8	Forced convection/diffusion elements(8-node)
	AC3D4	Acoustic elements(4-node linear tetrahedron)
	AC3D6	Acoustic elements(6-node linear triangular prism)
	AC3D8	Acoustic elements(8-node linear brick)
	AC3D10	Acoustic elements(10-node quadratic tetrahedron)
	AC3D15	Acoustic elements(15-node quadratic triangular prism)
	AC3D20	Acoustic elements(20-node quadratic brick)

Type of dimension	Type of element	Description
2D	DC2D3	Diffusive heat transfer or mass diffusion elements(3-node linear)
	DC2D4	Diffusive heat transfer or mass diffusion elements(4-node linear)
	DC2D6	Diffusive heat transfer or mass diffusion elements(6-node quadratic)
	DC2D8	Diffusive heat transfer or mass diffusion elements(8-node biquadratic)
	DCC2D4	Forced convection/diffusion elements(4-node)
	AC2D3	Acoustic elements(3-node linear)
	AC2D4	Acoustic elements(4-node bilinear)
	AC2D6	Acoustic elements(6-node quadratic)
	AC2D8	Acoustic elements(8-node biquadratic)
Shell	S3	Stress/displacement elements(3-node triangular general-purpose shell, finite membrane strains)
	S4	Stress/displacement elements(4-node general-purpose shell, finite membrane strains)
	S8	Stress/displacement elements(8-node doubly curved thick shell)
	DS3	Heat transfer elements(3-node triangular shell)
	DS4	Heat transfer elements(4-node quadrilateral shell)
	DS6	Heat transfer elements(6-node triangular shell)
	DS8	Heat transfer elements(8-node quadrilateral shell)
	ACIN3D3	Acoustic elements(3-node linear, acoustic infinite triangular element)
	ACIN3D4	Acoustic elements(4-node linear, acoustic infinite quadrilateral element)
	ACIN3D6	Acoustic elements(6-node quadratic, acoustic infinite triangular element)
	ACIN3D8	Acoustic elements(8-node quadratic, acoustic infinite quadrilateral element)

11-2-2. JMAG Magnetic Field Analysis, Iron Loss Analysis > Abaqus Heat Transfer Analysis

The Joule loss or iron loss calculated from a magnetic field analysis (static analysis, transient response analysis, frequency response analysis) or an iron loss analysis in JMAG is used in an Abaqus heat transfer analysis

11-2-2-1. Flow of Analysis

The following is the flow of analysis.

1. **Calculate a Joule loss by running a magnetic field analysis or an iron loss in an iron loss analysis in JMAG.**
2. **Prepare an input file for Abaqus.**
3. **Map the Joule loss or iron loss in step 1 to the input file for Abaqus in step 2 by using the [Heat Generation Mapping Tool For Abaqus] in JMAG.**
4. **Run a heat transfer analysis in Abaqus by using the file in step 3.**

11-2-2-2. Data transferred between programs and where it is received

- JMAG Magnetic field analysis Joule loss > Abaqus Heat transfer analysis
- JMAG Iron loss analysis Iron loss > Abaqus Heat transfer analysis
- JMAG Magnetic field analysis Joule loss + JMAG Iron loss analysis Iron loss > Abaqus Heat transfer analysis

11-2-2-3. Important Notes Regarding the Use of This Function

- Mapping of the heat generation distribution is supported.
  - The force is mapped using the of reference interval of electromagnetic force specified in the magnetic field analysis (frequency response analysis) to refer.
  - The force is mapped using the of reference interval of electromagnetic force specified in the magnetic field analysis (transient response analysis) to refer.
- Models in JMAG and Abaqus that have a different dimensions or divisions are supported.

11-2-2-4. JMAG Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

- Set partial model conditions for a partial model. Only the same area as the heat transfer analysis is mapped when the partial model condition is not set.

11-2-2-5. Procedures for Heat Generation Mapping Tool For Abaqus

1. **Select [Tools] > [Heat Generation Mapping Tool for Abaqus] from the menu bar of JMAG-Designer.**

   The [Heat Generation Mapping Tool for Abaqus] dialog box is displayed.

2. **Click [Browse] next to [Abaqus Input File], and then select the Abaqus file (*.inp) for mapping the heat generation.**

3. **Click [Browse] next to [Output File], and then enter the name and folder to save the Abaqus file (*.inp) after mapping.**

4. **Set the units (length, loss density, heat flux) in [Abaqus Units]**

5. **Click the [Mag Results] tab to map the Joule loss obtained in the magnetic field analysis.**

6. **Click the [Iron Loss Results] tab to map the iron loss results obtained in an iron loss analysis.**

   The following settings can be specified if the [Mag Results] tab is clicked in step 5.

   - [Use Mag Results] : Enables to use the magnetic field analysis results.
   - [Thermal Analysis Model]
     - [Target]: Displays a list of the sets specified in the Abaqus input file. Select the sets or surfaces to map the heat generation.

       All of the sets in the list can be selected or cleared using the [Selection Of All/ Non-Selection of All] check box.
     - [Scaling Factor]: Specifies the scaling factor for the heat generation to map. The value multiplied by the scaling factor is mapped.

- [Magnetic Field Analysis Model]
  - [Results File]: Specifies the JPLOT file for the JMAG magnetic field analysis.
  - [Average Loss]: Specifies transient magnetic field analysis results.

    Select the step (time) to average. If the time entered does not correspond to any step, the loss value obtained by linearly interpolating the loss between the proximate steps will be used.

    Select the [Use Rotation Average] check box to average the loss data in the circumferential direction for a model that has rotation motion.

    Do not select [Use Rotation Average] for a model that has translation motion.

    MEMO For results of 2D models or axisymmetric models, the rotation average cannot be used.
  - [Step Specification]: Specifies a specific step for reference. Enter the step number in the text box.
  - [Loss Generation Area]: Specifies the range to map the loss data from a 2D magnetic field analysis model in the Z-axis direction of a 3D thermal analysis model. Usually the face is Z=0.

7. Specify the following settings if the [Iron Loss Results] tab is clicked in step 5.

- [Use Iron Loss Results]: Enables to use the iron loss analysis results.
- [Thermal Analysis Model]
  - [Target]: Displays a list of the sets specified in the Abaqus input file. Select the sets or surfaces to map the heat generation.

    All of the sets in the list can be selected or cleared using the [Selection Of All/Non-Selection of All] check box.
  - [Scaling Factor]: Specifies the scaling factor for the heat generation to map. The value multiplied by the scaling factor is mapped.
- [Magnetic Field Analysis Model]
  - [Results File]: Specifies the JPLOT file for the JMAG iron loss analysis.
  - [Loss Type]: Specifies [Iron Loss], [Hysteresis Loss], or [Joule Loss] for the type of loss to map.
  - [Loss Generation Area]: Specifies the range to map the loss data from a 2D iron loss analysis model in the Z-axis direction of a 3D thermal analysis model. Usually the face is Z=0.

8. Click [Run].

The mapping starts.

11-2-2-6. Abaqus Heat Transfer Analysis Settings and Important Notes Regarding the Use of This Function

- Analysis steps for "Heat Transfer" need to be defined in the Abaqus input file to map.

■ Element types that can be used in Abaqus

The following are element types that can be used in Abaqus when it is linked with JMAG.

Type of dimension	Type of element	Description
3D	C3D4	Stress/displacement elements(4-node linear tetrahedron)
	C3D6	Stress/displacement elements(6-node linear triangular prism)
	C3D8	Stress/displacement elements(8-node linear brick)
	C3D10	Stress/displacement elements(10-node quadratic tetrahedron)
	C3D15	Stress/displacement elements(15-node quadratic triangular prism)
	C3D20	Stress/displacement elements(20-node quadratic brick)
	DC3D4	Diffusive heat transfer or mass diffusion elements(4-node linear tetrahedron)
	DC3D6	Diffusive heat transfer or mass diffusion elements(6-node linear triangular prism)
	DC3D8	Diffusive heat transfer or mass diffusion elements(8-node linear brick)
	DC3D10	Diffusive heat transfer or mass diffusion elements(10-node quadratic tetrahedron)
	DC3D15	Diffusive heat transfer or mass diffusion elements(15-node quadratic triangular prism)
	DC3D20	Diffusive heat transfer or mass diffusion elements(20-node quadratic brick)
	DCC3D8	Forced convection/diffusion elements(8-node)
	AC3D4	Acoustic elements(4-node linear tetrahedron)
	AC3D6	Acoustic elements(6-node linear triangular prism)
	AC3D8	Acoustic elements(8-node linear brick)
	AC3D10	Acoustic elements(10-node quadratic tetrahedron)
	AC3D15	Acoustic elements(15-node quadratic triangular prism)
	AC3D20	Acoustic elements(20-node quadratic brick)

Type of dimension	Type of element	Description
2D	DC2D3	Diffusive heat transfer or mass diffusion elements(3-node linear)
	DC2D4	Diffusive heat transfer or mass diffusion elements(4-node linear)
	DC2D6	Diffusive heat transfer or mass diffusion elements(6-node quadratic)
	DC2D8	Diffusive heat transfer or mass diffusion elements(8-node biquadratic)
	DCC2D4	Forced convection/diffusion elements(4-node)
	AC2D3	Acoustic elements(3-node linear)
	AC2D4	Acoustic elements(4-node bilinear)
	AC2D6	Acoustic elements(6-node quadratic)
	AC2D8	Acoustic elements(8-node biquadratic)
Shell	S3	Stress/displacement elements(3-node triangular general-purpose shell, finite membrane strains)
	S4	Stress/displacement elements(4-node general-purpose shell, finite membrane strains)
	S8	Stress/displacement elements(8-node doubly curved thick shell)
	DS3	Heat transfer elements(3-node triangular shell)
	DS4	Heat transfer elements(4-node quadrilateral shell)
	DS6	Heat transfer elements(6-node triangular shell)
	DS8	Heat transfer elements(8-node quadrilateral shell)
	ACIN3D3	Acoustic elements(3-node linear, acoustic infinite triangular element)
	ACIN3D4	Acoustic elements(4-node linear, acoustic infinite quadrilateral element)
	ACIN3D6	Acoustic elements(6-node quadratic, acoustic infinite triangular element)
	ACIN3D8	Acoustic elements(8-node quadratic, acoustic infinite quadrilateral element)

11-3. One-way Coupled Analysis (Abaqus > JMAG)

Runs an analysis using JMAG with results by Abaqus (stress).

■ Types of analyses

- Abaqus structural analysis > JMAG Magnetic Field Analysis, Iron loss Analysis

■ Required solver

- JMAG
  - Magnetic field analysis (one of the following)
    - Frequency response analysis (FQ)
    - Transient response analysis (TR, DP)
    - Iron loss calculation (LS)
- Abaqus
  - Structural analysis

11-3-1. Abaqus Structural Analysis (stress) > JMAG Magnetic Field Analysis, Iron Loss Analysis

The stress calculated from a structural analysis in Abaqus is used in a magnetic field analysis or an iron loss analysis.

11-3-1-1. Flow of Analysis

1. **Calculate the stress by running a structural analysis in Abaqus.**
2. **Set materials with magnetic properties that are stress dependent and stress distribution condition in a magnetic field analysis in JMAG.**
3. **Run a magnetic field analysis in JMAG.**

11-3-1-2. Data transferred between programs and where it is received

- Abaqus Structural analysis Stress > JMAG Stress dependent material, Stress distribution condition

11-3-1-3. Abaqus Structural Analysis Settings and Important Notes Regarding the Use of This Function

- Abaqus 6.10-2 needs to be installed on the calculation machine.
- For Windows, the folder Abaqus is installed is automatically detected.
- For Linux, the following path needs to be specified in LD_LIBRARY_PATH.

  Abaqus/6.10-2/exec/lbr

  Abaqus/6.10-2/External

  MEMO Set the environment variables by following the procedures below if the installation folder for Abaqus is not automatically detected in Windows. (for Windows XP)
  1) Select [Control Panel] in the [Start] menu of Windows.
  The [Control Panel] dialog box is displayed.
  2) Double-click [System].
  The [System Properties] dialog box is displayed.
  3) Select [Advanced] > [Environment Variables].
  The [Environment Variables] dialog box displayed.
  4) Select [New] under [User variables].
  The [New User Variable] dialog box is displayed.
  5) Enter [ABAQUSDIR] in the [Variable name] dialog box.
  6) Enter the path (example: C:\SIMULIA\Abaqus\6.10-2\) for the Abaqus installation folder in the [Variable value] box.
  Confirm that "\" is entered at the end of the path.
  7) Click [OK].
  8) Select [OK] in the [Environment variables] dialog box.
  9) Select [OK] in the [System Properties] dialog box.
  10) Close the [Control Panel] dialog box.

■ Element types that can be used in Abaqus

- The following are element types that can be used in Abaqus when it is linked with JMAG.

Type of dimension	Type of element	Description
3D	C3D4	Stress/displacement elements(4-node linear tetrahedron)
	C3D6	Stress/displacement elements(6-node linear triangular prism)
	C3D8	Stress/displacement elements(8-node linear brick)
	C3D10	Stress/displacement elements(10-node quadratic tetrahedron)
	C3D15	Stress/displacement elements(15-node quadratic triangular prism)
	C3D20	Stress/displacement elements(20-node quadratic brick)
	C3D27	Stress/displacement variable node elements(21 to 27-node brick)
2D	CPE3	Plane strain elements(3-node linear)
	CPE4	Plane strain elements(4-node bilinear)
	CPE6	Plane strain elements(6-node quadratic)
	CPE8	Plane strain elements(8-node biquadratic)
	CPS3	Plane stress elements(3-node linear)
	CPS4	Plane stress elements(4-node bilinear)
	CPS6	Plane stress elements(6-node quadratic)
	CPS8	Plane stress elements(8-node biquadratic)

11-3-1-4. JMAG Analysis Settings and Important Notes Regarding the Use of This Function

- Important notes on referring to an OBD file for Abaqus is the same as when referring to a PLOT file for JMAG and a JPLOT.

▶▶ Reference

For more information, see P.810 "10-6-3. Structural Analysis => Magnetic Filed Analysis, Iron Loss Analysis".

11-4. Two-way Coupled Analysis (JMAG < > Abaqus)

■ Types of analyses

- JMAG magnetic field analysis > Abaqus complete coupled analysis for thermal and stress
- JMAG magnetic field analysis > Abaqus dynamic analysis

■ Required solver

- JMAG
  - Magnetic field analysis (one of the following)
    - Static analysis (ST)
    - Frequency response analysis (FQ)
    - Transient response analysis (TR, DP)
- Abaqus
  - Structural analysis
    - Dynamic Temperature-displacement (coupled thermal and stress analysis)
    - Dynamics (dynamic analysis)

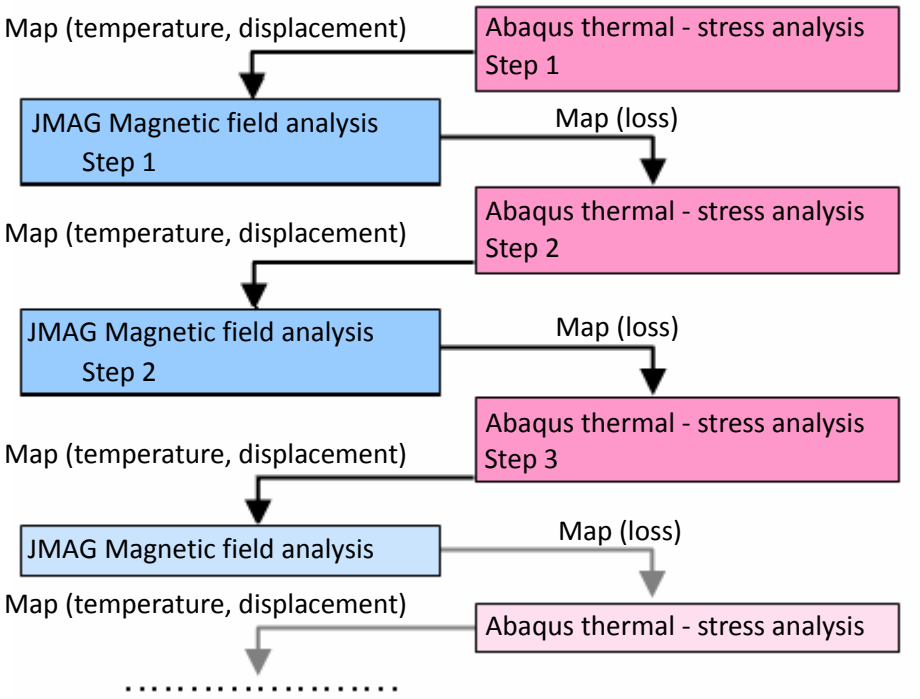
11-4-1. JMAG Magnetic Field Analysis (Joule Loss Density) < > Abaqus Fully Coupled Analysis for Thermal and Stress (Displacement/Temperature)

An analysis is run in Abaqus by using a result in JMAG (Joule loss density) and an analysis is run in JMAG by using a result in Abaqus (displacement, temperature). This function is useful for an induction heating analysis.

■ Processing procedures

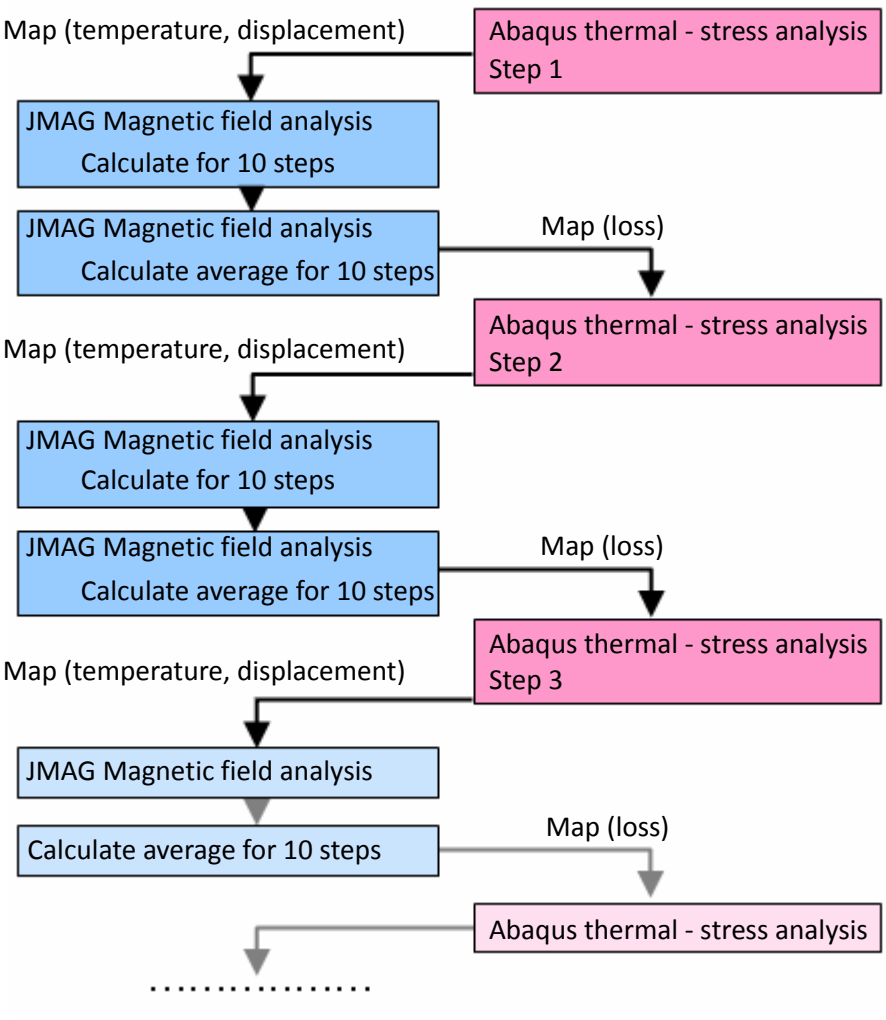
The coupled analysis is run by the following process.

Example 1: JMAG magnetic field analysis (static analysis, frequency response analysis) and Abaqus coupled thermal - stress analysis


Map (temperature, displacement)
Abaqus thermal - stress analysis Step 1
JMAG Magnetic field analysis Step 1
Map (loss)
Abaqus thermal - stress analysis Step 2
Map (temperature, displacement)
JMAG Magnetic field analysis Step 2
Map (loss)
Abaqus thermal - stress analysis Step 3
Map (temperature, displacement)
JMAG Magnetic field analysis
Map (loss)
Map (temperature, displacement)
Abaqus thermal - stress analysis

Example 2: JMAG magnetic field analysis (transient response analysis) and Abaqus thermal - stress analysis link (average loss is used)

(magnetic field analysis is run on 10 steps)


Map (temperature, displacement)
Abaqus thermal - stress analysis Step 1
JMAG Magnetic field analysis Calculate for 10 steps
JMAG Magnetic field analysis Calculate average for 10 steps
Map (loss)
Abaqus thermal - stress analysis Step 2
Map (temperature, displacement)
JMAG Magnetic field analysis Calculate for 10 steps
JMAG Magnetic field analysis Calculate average for 10 steps
Map (loss)
Abaqus thermal - stress analysis Step 3
Map (temperature, displacement)
JMAG Magnetic field analysis
Calculate average for 10 steps
Map (loss)
Abaqus thermal - stress analysis

11-4-1-1. Flow of Analysis

The following is the flow of analysis.

1. **Prepare an input file for a magnetic field analysis in JMAG.**
2. **Prepare an input file for a fully coupled thermal and stress analysis in Abaqus.**
3. **Run a two-way coupled analysis for JMAG and Abaqus from JMAG-Scheduler.**

11-4-1-2. Data transferred between programs and where it is received

- JMAG magnetic field analysis Joule loss > Fully coupled thermal and stress analysis
- JMAG Iron loss analysis Iron loss > Fully coupled thermal and stress analysis in Abaqus
- JMAG magnetic field analysis Joule loss + JMAG Iron loss analysis Iron loss > Fully coupled thermal and stress analysis
- Abaqus Fully coupled thermal and stress analysis Temperature => JMAG Magnetic field analysis, Temperature dependent material, Temperature load condition
- Abaqus Fully coupled thermal and stress analysis > JMAG Magnetic field analysis condition

11-4-1-3. Important Notes Regarding the Use of This Function

- This couple analysis is run from JMAG-Scheduler.

  ▶▶ Reference

  For more information about JMAG-Scheduler, see "Chapter 40. Running the Analysis," in the JMAG-Designer User's Manual.

- Align the geometry in the parts for JMAG and Abaqus. Aligning the geometry of the mesh is not required.
- This function is for when the dimensions in the model are different in JMAG and Abaqus, the division of full model or partial model is different, and difference in mesh geometry.
- The time it takes for an analysis depends on the settings in the displacement analysis in Abaqus.
- Run only the fully coupled thermal and stress analysis for an analysis procedure in Abaqus. This function is not supported for multiple types of analysis procedures.
- Specify "JMAG_" in front of the element set name in [Section Assignments] for parts that mapped physical quantities.
- An axisymmetric model needs to be used in JMAG and Abaqus when using an axi-symmetric model for this function.
- This function is not supported when one of model is an axisymmetric model and the other is of different dimension (3D or 2D).
- Abaqus6.10-2 needs to be installed into the calculation machine.
  - The installation folder for Abaqus is automatically detected on a Windows machine.
  - The following path needs to be set for LD_LIBRARY_PATH for Linux machines.
    - Abaqus/6.10-2/exec/lbr

- Abaqus/6.10-2/External

MEMO Set the environment variable when the installation folder for Abaqus is not automatically detected in Windows. For more information about the procedures, see P.836 "11-3-1. Abaqus Structural Analysis (stress) > JMAG Magnetic Field Analysis, Iron Loss Analysis".

11-4-1-4. JMAG Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

■ Model geometry

- The input file can either be a geometry data or a mesh data.

■ Material

- Creates a material with temperature dependent properties that is set to a part.
  - Set a temperature - electric conductivity curve as an electric property for material settings when using temperature dependency for electric conductivity.
  - Set a curve for temperature - relative permeability when setting the materials to use the temperature dependency of relative permeability.
  - Set a curve for temperature - magnetic susceptibility when using the temperature dependency for a magnetic property defined by a B-H curve (magnetic field - magnetic flux density curve).

▶▶ Reference

For more information, see P.20 "2-3. Magnetizing Properties of a Soft Magnetic Material" and P.33 "2-4-3. Specifying Temperature-dependent Electric Conductivity/Electric Resistivity (Magnetic Field Analysis, Electric Field Analysis)".

■ Study Properties

- An air region cannot be created when a direct solution method (FEM+BEM) is selected as the calculation method.
- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - An air region cannot be created when the [Only remesh air region] check box is selected.
- Select the [Joule Loss Density] check box.
  - Select the [Joule Loss Density] check box in the [Output Control] screen of [Study Properties] for JMAG-Designer.

■ Condition

- Specify the temperature distribution condition for parts with material properties that have temperature dependency. Select [Two Way] as the coupling type.
- Specify the heat source condition when transferring the iron loss from the FEM coil or the resistor component to the thermal analysis.
- The full model conversion in a magnetic field analysis and the circuit conversion needs to be enabled when the magnetic field analysis is from a partial model and the heat source condition is used. These settings are specified in the study properties for JMAG-Designer.
- The displacement condition is set and [Two Way] is selected as the [Coupling Type] when using the displacement.

  Important **The model can be deformed by using the displacement condition where parameters such as the magnetization direction cannot be modified.**
- The displacement condition can be used with the temperature distribution condition.
- The time information that is set in an Abaqus displacement analysis is used when using the time dependency condition.
  - A coupled analysis using the temperature and time can be run when a power supply for a time dependency is used.

11-4-1-5. Abaqus Fully Coupled Thermal and Stress Analysis Important Notes Regarding the Use of This Function

■ Model geometry

- A mesh data needs to be prepared for an input file.

■ Element types that can be used in Abaqus

The following are element types that can be used in Abaqus when it is linked with JMAG.

Type of dimension	Type of element	Description
3D	C3D4	Stress/displacement elements (4 nodes, linear, tetrahedron)
	C3D6	Stress/displacement elements (6 nodes, linear, triangular prism)
	C3D8	Stress/displacement elements (8 nodes, linear, brick)
	C3D10	Stress/displacement elements (10-nodes, quadratic, tetrahedron)
	C3D15	Stress/displacement elements (15-nodes, quadratic triangular, prism)
	C3D20	Stress/displacement elements (20-nodes, quadratic, brick)
2D	CPS3	Plane strain elements (3-node, linear)
	CPS4	Plane strain elements (4-nodes linear)
	CPS6	Plane strain elements (6-nodes, quadratic)
	CPS8	Plane strain elements (8-nodes, quadratic)
	CPE3	Plane stress elements (3-nodes, linear)
	CPE4	Plane stress elements (4-nodes, linear)
	CPE6	Plane stress elements (6-nodes, quadratic)
	CPE8	Plane stress elements (8-nodes, quadratic)
Axisymmetric	CAX3	Axisymmetric elements (3-nodes, linear triangular prism)
	CAX4	Axisymmetric elements (4-nodes, linear, quadrilateral prism)
	CAX6	Axisymmetric elements (6-nodes, quadratic, triangular prism)
	CAX8	Axisymmetric elements (8-nodes, quadratic, tetrahedron)

11-4-1-6. Coupled Analysis Settings and Important Notes Regarding the Use of This Function (JMAG-Scheduler)

■ Temperature evaluation position, Loss region

- The [Position] and [Loss Region] is specified for a 2D model in JMAG. Specify the distance of the Z-axis direction based on the XZ plane in [Position].
- Specify [Position] when the model is axisymmetric in JMAG. Specify the angle for the circumferential direction in the Y center point of axis based on the XY plane in [Position].

MEMO The results for an axisymmetric model in JMAG is mapped on a 3D model as 360 degrees in a circumferential direction. Therefore, setting the loss generation region is not required.

■ Average loss

- The loss for magnetic field analysis is averaged based on the trapezoidal rule when the setting for average loss selected.

■ Unit used in an input file for Abaqus

- Select the unit that is going to be used in an input file for Abaqus for [Length], [Loss Density], [Force], and [Force Density].

■ Map items and model conversion

- The Joule loss for a magnetic field analysis mapped in an input file for Abaqus or the iron loss in an iron loss analysis can be used when an iron loss condition is set in a magnetic field analysis for JMAG. The value with Joule loss and iron loss added can also be mapped.
- An analysis result of 1m in thickness is exported for a 2D analysis in JMAG. Therefore, the magnetic analysis result needs to be converted when JMAG and Abaqus models are 2D. Specify a value in [Scale Factor].
- Specify a height in [Enlargement Distance] for an Abaqus model when the model for JMAG is 2D and the model for Abaqus is 3D. The extended direction of the global coordinate system is the Z-axis.

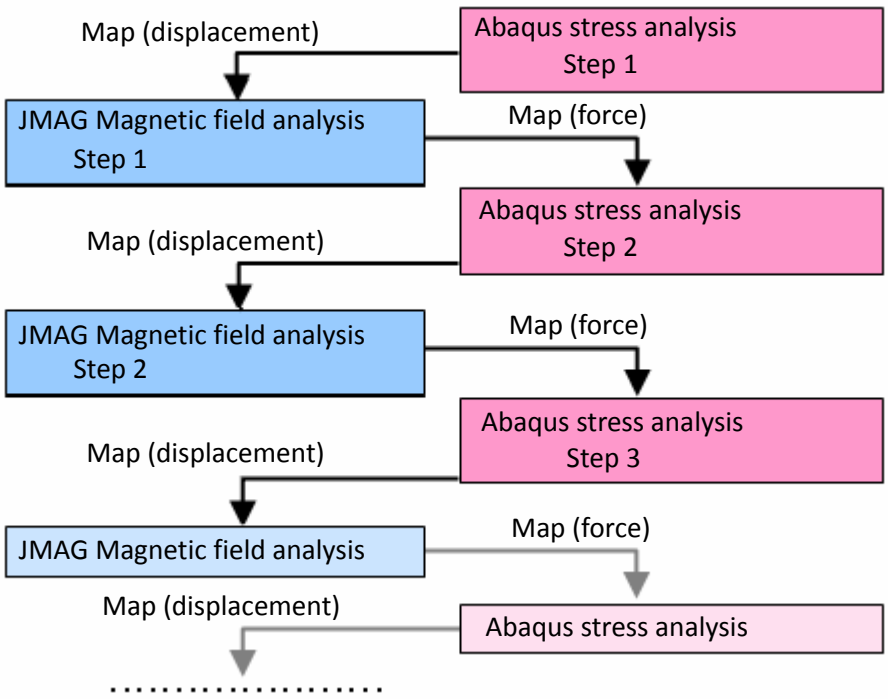
11-4-2. JMAG Magnetic Field Analysis (Force) < > Abaqus Dynamic Analysis (Displacement)

An analysis run by Abaqus uses results (force) from JMAG. Also, an analysis run by JMAG uses results from Abaqus (displacement).

■ Processing procedures

The coupled analysis is run by the following process.

Example: Coupled JMAG magnetic field analysis and Abaqus stress analysis



11-4-2-1. Flow of Analysis

The following is the flow of analysis.

1. **Prepare an input file for a magnetic field analysis in JMAG.**
2. **Prepare a dynamic analysis input file for Abaqus.**
3. **Run a two-way coupled analysis for JMAG and Abaqus from JMAG-Scheduler.**

11-4-2-2. Data transferred between programs and where it is received

- JMAG Magnetic Field Analysis (Force) > Abaqus Dynamic Analysis
- Abaqus Dynamic analysis Displacement > JMAG Magnetic Field Analysis Displacement condition

11-4-2-3. Important Notes Regarding the Use of This Function

- This couple analysis is run from JMAG-Scheduler.

  ▶▶ Reference

  For more information about JMAG-Scheduler, see "Chapter 40. Running the Analysis," in the JMAG-Designer User's Manual.
- Align the geometry in the parts for JMAG and Abaqus. Aligning the geometry of the mesh is not required.
- This function is for when the dimensions in the model are different in JMAG and Abaqus, the division of full model or partial model is different, and difference in mesh geometry.
- The time it takes for an analysis depends on the settings in the displacement analysis in Abaqus.
- Run only the dynamic analysis for an analysis procedure in Abaqus. This function is not supported for multiple types of analysis procedures.
- Specify "JMAG_" in front of the element set name in [Section Assignments] for parts that mapped physical quantities.
- An axisymmetric model needs to be used in JMAG and Abaqus when using an axisymmetric model for this function. This function is not supported when one model is an axisymmetric model and the other is of different dimension (3D and 2D).
- Abaqus6.10-2 needs to be installed into the calculation machine.
  - The installation folder for Abaqus is automatically detected on a Windows machine.
  - The following path needs to be set for LD_LIBRARY_PATH for Linux machines.
    - Abaqus/6.10-2/exec/lbr
    - Abaqus/6.10-2/External

MEMO Set the environment variable when the installation folder for Abaqus is not automatically detected in Windows. For more information about the procedures, see P.836 "11-3-1. Abaqus Structural Analysis (stress) > JMAG Magnetic Field Analysis, Iron Loss Analysis".

11-4-2-4. JMAG Magnetic Field Analysis Settings and Important Notes Regarding the Use of This Function

■ Model geometry

- The input file can either be a geometry data or a mesh data.

■ Study Properties

- An air region cannot be created when a direct solution method (FEM+BEM) is selected as the calculation method.
- Select the two-way coupled analysis.
  - Select the [Two way coupled analysis] check box in the [Coupled Analysis] screen for JMAG-Designer.
  - An air region cannot be created when the [Only remesh air region] check box is selected.

■ Condition

- The displacement condition is set and [Two Way] is selected as the [Coupling Type] when using the displacement.

  Important **The model can be deformed by using the displacement condition but parameters such as the magnetization direction cannot be modified.**
- Use the nodal force or Lorentz force when setting the electromagnetic force condition. This function is not supported for surface force.

11-4-2-5. Abaqus Dynamic Settings and Important Notes Regarding the Use of This Function

■ Model geometry

- A mesh data needs to be prepared for an input file.

■ Element types that can be used in Abaqus

The following are element types that can be used in Abaqus when it is linked with JMAG.

Type of dimension	Type of element	Description
3D	C3D4	Stress/displacement elements (4 nodes, linear, tetrahedron)
	C3D6	Stress/displacement elements (6 nodes, linear, triangular prism)
	C3D8	Stress/displacement elements (8 nodes, linear, brick)
	C3D10	Stress/displacement elements (10-nodes, quadratic, tetrahedron)
	C3D15	Stress/displacement elements (15-nodes, quadratic triangular, prism)
	C3D20	Stress/displacement elements (20-nodes, quadratic, brick)
2D	CPS3	Plane strain elements (3-nodes, linear)
	CPS4	Plane strain elements (4-nodes linear)
	CPS6	Plane strain elements (6-nodes, quadratic)
	CPS8	Plane strain elements (8-nodes, quadratic)
	CPE3	Plane stress elements (3-nodes, linear)
	CPE4	Plane stress elements (4-nodes, linear)
	CPE6	Plane stress elements (6-nodes, quadratic)
	CPE8	Plane stress elements (8-nodes, quadratic)
Axisymmetric	CAX3	Axisymmetric elements (3-nodes, linear triangular prism)
	CAX4	Axisymmetric elements (4-nodes, linear, quadrilateral prism)
	CAX6	Axisymmetric elements (6-nodes, quadratic, triangular prism)
	CAX8	Axisymmetric elements (8-nodes, quadratic, tetrahedron)

11-4-2-6. Coupled Analysis Settings and Important Notes Regarding the Use of This Function (JMAG-Scheduler)

■ Unit used in an input file for Abaqus

- Select the unit that is going to be used in an input file for Abaqus for [Length], [Force], and [Force Density].

■ Map items and model conversion

- An analysis result of 1m in thickness is exported for a 2D analysis in JMAG. Therefore, the magnetic analysis result needs to be converted when JMAG and Abaqus models are 2D. Specify a value in [Scale Factor].
- Specify a height in [Enlargement Distance] for an Abaqus model when the model for JMAG is 2D and the model for Abaqus is 3D. The extended direction of the global coordinate system is the Z-axis.

Chapter 12. Tools

This chapter describes the tools included in JMAG.

12-1. Listing of Tools

■ JMAG-Express (JMAG-Studio motor template)

Creates 2D motor models.

MEMO The motor template tool is now merged with JMAG-Express in JMAG-Designer Version 11.0.

▶▶ Reference

For more information about JMAG-Express, see "JMAG-Designer User's Manual."

■ Parametric Analysis

Gives the results for multiple cases for the same model by applying different values to the condition and geometry parameters.

▶▶ Reference

For more information about Parametric Analysis, see "JMAG-Designer User's Manual" or "JMAG-Studio User's Manual."

■ JMAG-RT

Converts the FEM analysis results obtained from JMAG into format readable by the circuit/control simulators.

■ Magnetization for Magnetic Field Analysis

Determines the magnetization direction for each component of a magnet based on vector data contained in the magnetic field analysis results.

■ Calculate Iron Loss

Calculates the iron loss by using the result file obtained by magnetic or electric field analyses.

■ Calculate Particle Trajectory

Calculates the flight orbit of charged particles in magnetic and electric fields by using the result file obtained by magnetic or electric field analyses.

■ Calculate Field Value

Calculates the magnetic flux density and electric fields at a specified position (in the air), using magnetization vectors, current densities, beam currents, and charge distribution data stored in the result file obtained by magnetic or electric field analyses.

■ Calculate Far Field

Calculates the electromagnetic field at points sufficiently far away from finite-element analysis regions by using the equivalent surface current at points specified by an analysis model.

■ Creating Averaged Loss (Power Loss File)

Creates a Joule-loss file for time-averaged thermal analysis by using joule-loss data stored in the result file obtained by magnetic field analysis.

■ Temperature for Structural Analysis

Adds the temperature data stored in a result file obtained by thermal analysis to structural analysis data.

■ Force Mapping for Structural Analysis

A file used in a structural analysis (static analysis/frequency response analysis) can be created by using electromagnetic force or magnetostriction data in each node included in a result for a magnetic field analysis.

■ Force Export for Virtual.Lab (JMAG-Designer)

The nodal force calculated from a magnetic field analysis (transient analysis) from JMAG is used for the vibration/noise analysis in Virtual Lab which is the general purpose electromagnetic optics design and evaluation program.

Force Mapping for Abaqus

Creates a file to use in the structural analysis of the nonlinear finite element analysis program Abaqus by using the electromagnetic force data obtained from the JMAG magnetic field analysis (static analysis/transient response analysis/frequency response analysis).

Heat Generation Mapping for Abaqus

Creates a file to use in the heat transfer analysis of the nonlinear finite element analysis program Abaqus by using the Joule loss or iron loss data obtained from the JMAG magnetic field analysis (static analysis/transient response analysis/frequency response analysis) and the iron loss analysis.

Exporting Multi-purpose Files (JMAG-Designer)

Mesh, analysis data (time, frequency), and result data (physical quantities) created in JMAG are exported in Nastran format.

Files that are outputted using this tool, can be used in simulation software other than JMAG.

Calculate Motor Inductance (For Three-phase Permanent Magnet Synchronous Motor)

Calculates the d-axis and q-axis inductance in a three-phase permanent magnet synchronous motor using a JCF file of magnetic field analysis.

Calculate Coil Inductance

Calculates the inductance in a model with coils, such as a transformer, using a JCF file of magnetic field analysis.

Calculate Busbar Inductance

In order to calculate the Inductance of a conductor, the inside of the conductor is divided into several parts, and the inductance of each part and the partial inductance between the parts are calculated.

Calculate Design Approximate

This is a function for building up an approximation equation from the combination of design variables and response variables obtained by the analysis. Based on this approximation equation, calculates the response variables from the arbitrary design variables.

Important **Some of the tools described herein may require separate licenses for each. For further details, see our website.**

Important **From JMAG-Studio 9.1, a feature equivalent to the Calculate Motor Inductance tool has been added to JMAG-RT. However, the following features of the Calculate Motor Inductance tool are not supported in JMAG-RT.**
- Setting of current by d-/q-axes
- Export of PSIM Lookup table file
- Option for saving result files (all result files will be saved after the calculation with JMAG-RT.)

The tools available in JMAG-Studio and JMAG-Designer are shown in the following table.

Tool	JMAG-Studio	JMAG-Designer
JMAG-Express	O	O
Parametric Analysis*	O	O
JMAG-RT	O	O
Magnetization for Magnetic Field Analysis*	O	O
Calculate Iron Loss*	O	O
Calculate Particle Trajectory	O	X
Calculate Field Value	O	X
Calculate Far Field	O	X
Creating Averaged Loss (Power Loss File)	O	X
Temperature for Structural Analysis	O	X
Force Mapping for Structural Analysis*	O	O
Force Export for Virtual.Lab.	X	O
Force Mapping for Abaqus	X	O
Heat Generation Mapping for Abaqus	X	O
Multi-purpose File Export	X	O
Calculate Motor Inductance	O	O
Calculate Coil Inductance	O	O
Calculate Busbar Inductance	O	O
Calculate Design Approximate	O	X
PLOT File Conversion	X	O

MEMO Tools displayed with an asterisk have the same functions in JMAG-Studio and JMAG-Designer, but differ in settings or procedures.

MEMO For more information about the parametric analysis, see “Chapter 14, Parametric Analysis Tool” in the JMAG-Studio User's Manual or “Chapter 42 Parametric Analysis” in the JMAG-Designer User's Manual.

MEMO For more information about setting magnetization to magnetic field analysis in JMAG-Studio, see this manual. For more information, see “Chapter 28, Assigning Materials and Setting Material Properties” in the JMAG-Designer User's Manual.

MEMO For more information about the iron loss calculation in JMAG-Studio, see this manual. For more information in JMAG-Designer, see “Chapter 26, Creating a Study by Selecting an Analysis Type,” “Chapter 29 Study Properties and Conditions of Magnetic Field Analysis,” and “Chapter 30 Study Properties and Conditions of Iron Loss Analysis” of the JMAG-Designer User's Manual.

MEMO For more information about setting the electromagnetic force for a structural analysis file, see either the JMAG-Studio User's Manual or the JMAG-Designer User's Manual.

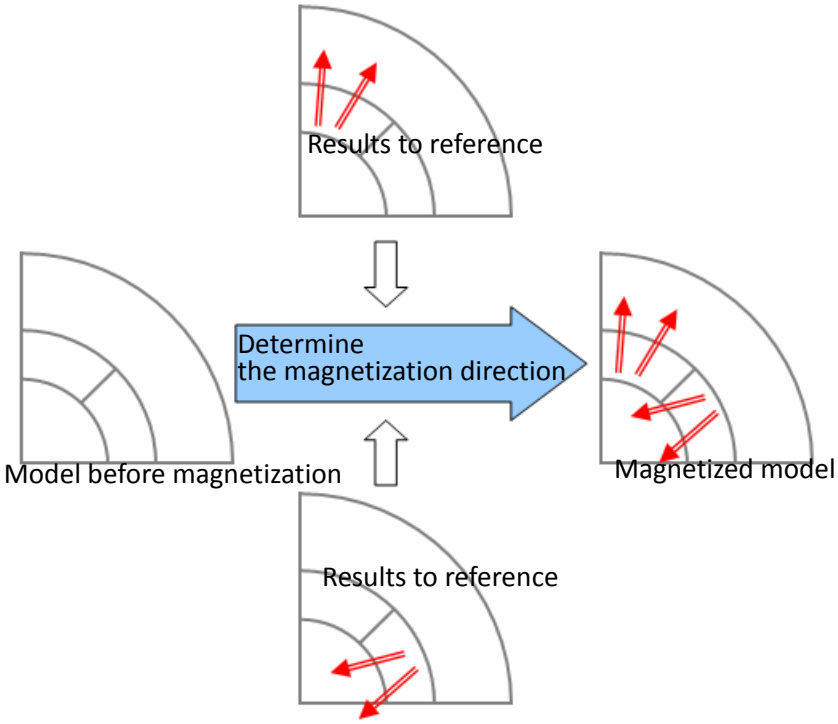
12-2. Magnetization for Magnetic Field Analysis

▶▶ Reference

For more information about magnetization procedures for JMAG-Designer, see the JMAG-Designer User's Manual, "Chapter 28, Assigning Materials and Setting Material Properties."

12-2-1. Function

The magnetization direction of each element of a permanent magnet is determined using this function. This task is performed using the vector data stored in the result file obtained from a magnetic field analysis.



Schematic diagram of magnetization analysis

For reference, this section uses the word "magnetization" in the sense that it determines the magnetization direction and coercive force of a permanent magnet and imparts permanent magnetism to the material properties of a magnet.

12-2-2. Procedures for Executing the Tool (JMAG-Studio)

1. **From the menu bar in JMAG-Studio, select [Tools] > [Magnetization for Magnetic Field Analysis].**

   The [Magnetization for Magnetic Field Analysis] dialog box will appear.

2. **Click [Add] in the [Plot Files] group box.**

   The [Reference Plot File] dialog box will appear.

3. **Click [Browse] in the [Reference Plot File] group box, and specify the PLOT file of the magnetic field analysis to refer to.**

4. **Click [Open].**

   The [Reference Plot File] dialog box will return.

5. **Select one of the following three options in the [Reference Data Type] group box.**

   - [Magnetic Flux Density]
   - [Magnetization]
   - [Magnetic Field]

   e.g.) If [Magnetic Flux Density] radio button is selected, the vector data of magnetic flux density is set for the magnetization.

6. **Select either of the two options in the [Value] group box, and enter the settings for the selected option.**

   - [Specified Step]: Specify directly the step ID to be referenced.

     After selecting the radio button, enter in the [Step] text box the step ID of the analysis result to be used. When referencing the PLOT file of a static magnetic-field analysis, enter “1” in the [Step] text box.
   - [Max. Value from Specified Step Range]: Specify the range by entering the start and end step IDs.

     After selecting the radio button, enter the value in [Step from] and [Step to].

7. Select the method to determine the magnitude of coercive force in a permanent magnet in the Magnetization Type group box.

▶▶ Reference

For details of the magnetization types, see the P.865 "12-2-3-1. Magnetization Type".

- [Complete Magnetization (Isotropic/Anisotropic)]: No setting is necessary in the [Magnetization Type] group box.
- [Incomplete (Isotropic)]: Specify the magnetic flux density dependency of magnetizing ratio or that of both the magnetizing ratio and recoil relative permeability by a point sequence.

  After selecting [Incomplete (Isotropic)], click the [Setting] button in the [Point Sequence] combo box. The magnetic flux density dependency of the following items can be specified by a point sequence.

  - Magnetization Ratio: If selecting this, set the following items.

    ```
    1: Magnetic Flux Density (T)
    2: Magnetization Ratio (0.0-1.0)
    ```

  - Magnetization Ratio and Recoil Relative Permeability: If selecting this, set the following items.

    ```
    1: Magnetic Flux Density (T)
    2: Magnetization Ratio (0.0-1.0)
    3: Recoil Relative Permeability
    ```

- [Incomplete (Anisotropic)]: Specify the magnetic flux density dependency of magnetizing ratio or that of both the magnetizing ratio and recoil relative permeability by a point sequence, and enter the value in the [Maximum Coercive Force (A/m)] text box.
- [User Defined]: Define the method used to determine the magnetization vector with the user subroutine *setusr*. Set the maximum coercive force and the user definition.

  After selecting the radio button, click [Setting] in the [Point Sequence] group box to open the [Point Sequence List] dialog box. The point sequence to be created here does not need to be a magnetizing point sequence.

  To set the maximum coercive force, enter the value in the [Maximum Coercive Force (A/m)] text box.

  A negative integer can be set for the [User Component (Negative Integer)]. The value set here will be set as *itypa* in a user subroutine.

  ▶▶ Reference

  For details of the user subroutine, see P.868 "12-2-4. User Subroutine setusr".

8. To use the transformation table, click [Add] in the [Transformation Table] group box to open the [Transformation Definition] dialog box, and enter the settings.

If this setting is not necessary, click [OK] to close the [Reference Plot File] dialog box and proceed to step 9.

If the transformation table is set, the coordinate values and vector data of the reference file can be rotated or transferred. Since the direction can be inverted again after rotation/translation, the model of a magnet array having their poles reversed alternately between neighboring magnets can be used.

To set the table, click [Add] of the [Transformation Table] to open the [Transformation Definition] dialog box. Select either option under [Type] and specify the settings required for the selected option.

- [Rotational]: Select [Periodic] or [Anti-Periodic] in the [Periodicity] group box, and enter the following items.
  - [X-coordinate (m)], [Y-coordinate (m)], and [Z-coordinate (m)] in the [Point on Rotation Axis] group box
  - [X], [Y], and [Z] in the [Rotation Axis/Direction Vector] group box
  - [Angle (deg)] in the [Rotation] group box
- [Translational]: Select [Periodic] or [Anti-Periodic] in the [Periodicity] group box, and enter the following items.
  - [X], [Y], and [Z] in the [Rotation Axis / Direction Vector] group box
  - [Distance (m)] in the [Translation] group box

When the setting is complete, click [OK], and then click [OK] in the [Reference Plot File] dialog box to return to the [Magnetization for Magnetic Field Analysis] dialog box.

Important **In the Transformation Table, multiple rotations and translations can be specified. In this, the magnetization will only be mapped at the final location determined by the transformations specified. Mapping cannot be done for each transformation individually.**

9. To have the magnet magnetized during the calculation, select the [Magnetize When Running Analysis] check box.

If this option is off, the model will be magnetized once you click [OK] to close the [Magnetizing Tool] dialog box.

10.Specify the file for magnetization and the file to be output after the magnetization.

If no specified, magnetization will be applied to the current model.

To select files, select the [Set to Existing File] check box. Click [Setting], and enter the following items in the [Select File] dialog box.

- [Input File Name]: Enter the file name, or click [Browse] to specify the file in the [Open] dialog box.
- [Output File Name]: Enter the file name, or click [Browse] to specify the file name in the [Save As] dialog box.

  After setting, click [OK] to return to the [Magnetization for Magnetic Field Analysis] dialog box.

At this point, the [Set to Existing File] check box is selected, and the [Magnetize When Running Analysis] check box is not available.

11.To execute magnetization calculation in the background, select the [Background] check box.

12.To display the status of a calculation, select the [Show Status] check box.

13.Click [Export] or [Load] if necessary.

- [Export]: To output the parameters to a file, click the [Export] button and open the [Save As] dialog box, and then enter the file name in the [File Name] text box and click [Save]. The data is save in a JCF file.
- [Load]: Click the [Load] button. The [Open] dialog box is shown. Specify the file in the list or text box, and then click [Open]. JCF files and parameter files for versions before JMAG-Studio 9.0 can be loaded.

14.Click [OK].

The [Magnetizing Tool] dialog box is closed.

If the [Magnetize When Running Analysis] check box is cleared in step 9, the magnet will be magnetized.

If you selected the [Magnetize When Running Analysis] check box in step 9, and you want to change the setting, select [Conditions] > [Modify Conditions] from the menu bar and open the [Condition List] dialog box. Select [Magnetizing Tool] from the list and click [Modify] to open the [Magnetizing Tool] dialog box, and re-enter the settings.

12-2-3. Important Notes Regarding the Use of This Function (JMAG-Studio)

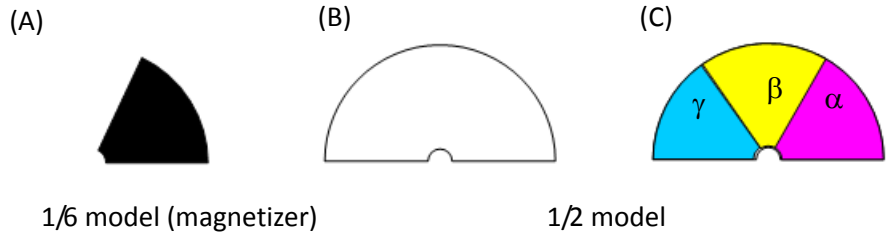
- Magnetizing is performed only on materials for which the [Use Magnetization Program] check box in the [Material] dialog box is selected when the material properties are set.

  ▶▶ Reference

  For more information about the [Use Magnetization Program] check box, see the JMAG-Studio User's Manual, “Chapter 9. Setting Conditions of Magnetic Field Analysis”, “Material Properties,” or “Creating a Magnet Material.”
- Multiple PLOT files can be referenced. In that case, files are used for magnetization starting from the top of the list, and will be overwritten sequentially by succeeding files.
- A search will be made to find the element of reference PLOT file (after the rotation or translation) that includes the center-of-mass coordinate of each element of a permanent magnet in a model to be magnetized. Then the magnetizing direction will be determined using the vector of the element searched.
- If a corresponding element is not found in the files to be referenced, magnetizing will not be executed. In that case, the value of the magnetizing vector will remain the same as that before magnetizing.
- When verifying the magnetizing direction of a permanent magnet after magnetizing, select the material ID of the permanent magnet in the conditions list displayed on the screen.
- In the setting for a permanent magnet after it has been magnetized, the magnetizing direction type will be transformed for each element. In addition, the check next to each material in the [Use Magnetization Program] check box will be cleared.
- Be sure not to create a mesh again for the magnetized model data if you mesh the model again using the [Semi-Automatic Mesh] option, the magnetization vector will become zero. Even if you use the [Automatic Mesh] or [Method in JMAG-Studio 9.0] option, it may lead to lowered calculation accuracy.
- If you need to generate mesh for the model again, make sure to magnetize the model after the mesh generation.
- When conducting magnetization for a partial model to which the patch mesh function is set, set [Magnetize When Running Analysis] to conduct magnetization during the analysis.

- Even if the number of divisions is different between the magnetizer model in a plot file and the model to be magnetized using the plot file, the magnetization can still be performed.

  For example, if a 1/2 model (B) is magnetized using a plot file containing a 1/6 model (A), perform the magnetization as follows.



1/6 model (magnetizer) 1/2 model

i. Click [Add] in the [Magnetization for Magnetic Field Analysis] dialog box, and then select the plot file A. Set parameters, and then click [OK]. With this setting, the region α in the figure (C) is magnetized.

ii. Click [Add] in the [Magnetization for Magnetic Field Analysis] dialog box, and select the plot file A, and then set parameters. Set the transformation table with rotation angle of 60 degrees, and then click [OK]. With this setting, the region β in the figure (C) is magnetized.

iii. Click [Add] in the [Magnetization for Magnetic Field Analysis] dialog box, and select the plot file A, and then set parameters. Set the transformation table with rotation angle of 120 degrees, and then click [OK]. With this setting, the region γ in the figure (C) is magnetized.

12-2-3-1. Magnetization Type

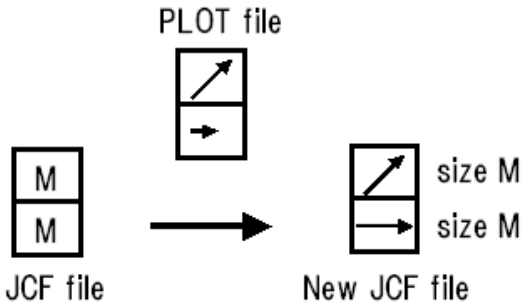
There are five magnetization types:

- Complete Magnetization (Isotropic, Anisotropic)
- Incomplete Magnetization (Isotropic)
- Incomplete Magnetization (Anisotropic)
- User defined

■ Complete Magnetization (Isotropic, Anisotropic)

Completely magnetize the permanent magnet. Applies the coercive force set for the magnet in the setting of materials.

Example: When only the direction of magnetic flux density is referenced from the PLOT file:



▶▶ Reference

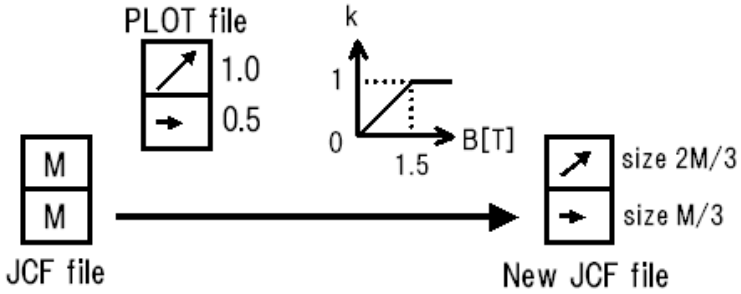
- For more information about the setting procedures for the magnetic coercive force, see the JMAG-Studio User's Manual, “Chapter 9. Setting Conditions of Magnetic Field Analysis.”
- For more information about the material properties, see P.78 "3-4. Material Property for Each Part: Permanent Magnet".

■ Incomplete Magnetization (Isotropic)

With this setting, the isotropic magnet is incompletely magnetized. For the coercive force of each element of the magnet, it applies the product of the magnetization ration determined by the flux density (or magnetic field, magnetization) generated in the elements in the reference PLOT file, and the coercive force (maximum value) specified in the Material dialog box.

The orientation is the same as flux density (or magnetic field, magnetization) generated in the elements in the reference PLOT file. The magnetization ratio is defined with a point sequence as the function of the magnetic flux density and recoil relative permeability.

Example) If using the orientation and magnitude in the PLOT file (magnetization ratio is obtained based on the magnitude).



■ Incomplete Magnetization (Anisotropic)

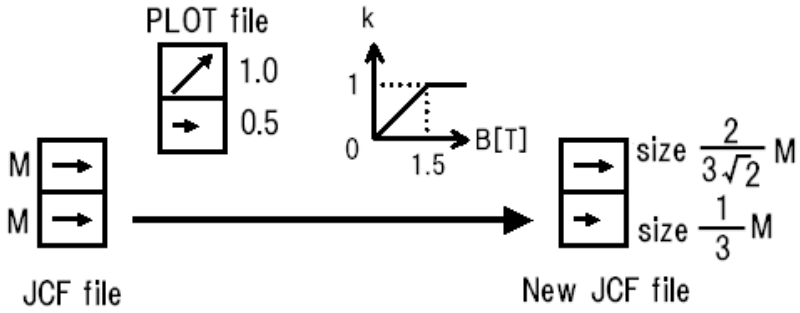
With this setting, the anisotropic magnet is incompletely magnetized. The orientation same as the magnetization victor already set on the model is used, allowing for the different orientations between the magnetization and magnetic field.

The magnetization ration is defined with a point sequence as the function of the magnetic flux density and recoil relative permeability. For the maximum coercive force, the value entered in the [Magnetizing Tool] dialog box is used.

Once magnetized using this tool, the magnetization direction corresponds to the orientation set on the model, and the coercive force is the product of the maximum coercive force entered specified in the Magnetizing Tool dialog box and the magnetization ratio of each element. If selecting Incomplete Magnetization (Anisotropic), the coercive force value entered in the Material dialog box is not used.

If selecting Incomplete Magnetization (Isotropic), the flux density vector in the PLOT file referenced for determining the magnetization ratio is used, whereas for Incomplete Magnetization (Anisotropic) uses the projective component of the flux density vector orientation in the referenced PLOT file.

Example) If using the orientation and magnitude in the PLOT file



References the magnetization vector in the original solver data file and the size of the projection component in the PLOT file, and uses the size data to obtain the magnetization ratio.

■ User Defined (*setusr*)

The method used to determine the magnetizing vector is defined by using with user subroutine *setusr*.

The point sequence, which is set when the [User Defined] radio button is selected, may not necessarily be a magnetizing point sequence. Furthermore, the maximum coercive force can be set. The maximum coercive force and point sequence are referenced in user subroutine *setusr*.

If a negative integer is entered into the [User Defined] text box, a reference PLOT file can be identified in the user subroutine *setusr* (868 page).

12-2-3-2. Transformation Table

Rotation (or translation) of the coordinate values and vector data included in the PLOT file can be done using the transformation table.

In the case of vector data, their direction can further be reversed after the movement. Therefore, a model of a magnet array with their poles reversed alternately between neighboring magnets can be dealt with.

If specifying multiple transformation tables, the transformation is carried out in the descending order.

12-2-4. User Subroutine *setusr*

12-2-4-1. Overview

The user subroutine *setusr* is used when defining the method used to determine the magnetization vector within the magnetizing tool to permanent magnet.

The magnetizing value of permanent magnet (*amagv*) is defined based on such as the orientation vector and magnetizing point sequence, which are passed on from the upper level subroutine. *setusr* is called for every referenced PLOT file in element unit within the permanent magnet.

Important **DLL files used in JMAG-Studio cannot be used in JMAG-Designer. The files need to be rebuilt in a new compiler.**

12-2-4-2. Arguments

Variable	Details	I / O
nplot	Total number of reference PLOT files	I
iplot	Current reference PLOT file number	I
itypa	Identification number of user subroutine (Negative integer entered in the dialog box)	I
ng	Order number of permanent magnet in input file	I
dirmg (3,maxmg)	Orientation vector (magnitude 1)	I
ampb (maxmag)	Magnitude of orientation vector	I
ifind (maxmg)	If there is an element in a PLOT file that coincides with an element of a permanent magnet in an input file, the variable is not 0, and if not, the variable is 0.	I
tablx (numpo)	X-component of magnetizing point sequence	I
tably (numpo)	Y-component of magnetizing point sequence	I

Variable	Details	I / O
numpo	Number of points in magnetizing point sequence	I
amagv(3,maxmg,num-mag)	Magnetization vector of permanent magnet	O
nmg (nummag)	Group ID of permanent magnet	I
magel (nummag)	Number of elements within permanent magnet	I
nemag(maxmg, num-mag)	Element ID of elements within permanent magnet	I
maxmg	Maximum element ID within permanent magnet group	I
numat	Number of material conditions in input file	I
iflag (numat)	Material type of input file	I
maxtp	Maximum point sequence number for material conditions in input file	I
ntab (numat)	Number of point sequence for material conditions in input file	I
tabl (numat)	Point sequence for material conditions in input file	I
cmu (numat)	Relative permeability of material conditions in input file (linear)	I
mtyp (numel)	Material ID of element in input file	I
numel	Number of elements in input file	I
amaxm	Maximum magnetization (Maximum magnetization value entered in the dialog box)	I

12-2-4-3. Example

The following is an example of magnetization using *setusr*.

In this example, the residual magnetic flux *Br* is determined from point sequence data (X-axis: amplitude of orientation magnetic field (A/m), Y-axis: residual magnetic flux (T)) defined by the magnetizing point sequence and the amplitude (A/m) of the orientation magnetic field of the magnetizing device. The retaining force *Hc* is obtained from the gradient of the demagnetization curve set for the magnet within the input data for *Br* and analysis before magnetization. Magnetization vectors are then assigned to each element.

```
      subroutine setusr(nplot,iplot,itypa,ng,dirmg,ampb,ifind,
     #   tablx,tably,numpo,
c           -- Permanent Magnet Data
     #      amagv,nmg,magel,nemag,maxmg,nummag,
c             -- Material Data
     #           numat,iflag,maxtp,ntab,tabl,cmu,mtyp,numel,
     #           amaxm)
c
      implicit real*8 (a-h,o-z)
c
      include '../comn/file.com'
      include '../comn/cntrl.com'
      include '../comn/number2.com'
      include '../comn/const.com'
c
      dimension dirmg(3,maxmg),ampb(maxng),ifind(maxmg),
    #        tablx(numpo),tably(numpo),
    #        amagv(3,maxmg,nummag),nmg(nummag),
    #        magel(nummag),nemag(maxmg,nummag),
    #
iflag(numat),ntab(numat),tabl(2*maxtp,numat),mtyp(numel)
      complex*16 cmu(numat)
c
      do 100 ne=1,magel(nmg(ng))
```

… Loops until the number of elements of permanent magnet.

```
c       -- get element ID
        nel=nemag(ne,nmg(ng))
```

… Gets the number of this element.

```
c       -- if corresponding element is find.
        if(ifind(ne).eq.1) then
```

… If there is a PLOT file element that coincides with a permanent magnet element.(Coincidence is decided by upper levelof subroutine.)

```
          do 1000 i=1,3
```

```
            amagv(i,ne,nmg(ng)) =dirmg(i,ne)*ampb(ne)
```

… Substitute temporary a magnetizing vector into “amagv”.

(Here the magnetizing vector is assumed to be a magnetic field.)

```
 1000   continue
          ampH = sqrt(amagv(1,ne,nmg(ng))**2
     #                   +amagv(2,ne,nmg(ng))**2
     #                   +amagv(3,ne,nmg(ng))**2)
```

… Determines the magnetic field force of this element.

```
          call hokan(tablx,tably,numpo,ampH,Br)
```

… Passes ampH to the subroutine hokan to get the remanent magnetic flux Br from point sequence defined by magnetizing dialog box.

```
c         -- material type ID. of this element
          mm=mtyp(nel)
```

… Gets the material ID of this element.

```
          if(iflag(mm).eq.1) then
```

… If the magnetization property of this element is linear.

```
c         -- Linear material
            Hc = Br / (cmu(mm)*pmu0)
```

… Gets coercive force Hc from Br and relative permeability.

```
          endif
c         -- Update amagv
          do 1010 i=1,3
            amagv(i,ne,nmg(ng)) = Hc * amagv(i,ne,nmg(ng)) /
ampH
```

…Taking vector with direction of magnetic field and magnitude Hc as magnetization vector, updates amagv.

```
 1010   continue
c
         endif
  100 continue
c
      return
c
      end
```

12-3. Calculate Iron Loss

▶▶ Reference

- For more information, see P.226 "3-21. Iron Loss Calculation"
- For more information, see “Iron Loss Calculation” in Chapter 30 “Study Properties and Conditions of Iron Loss Analysis” in the JMAG-Designer User's Manual.

12-3-1. Function

This tool calculates the loss density distribution and the loss amount in the specified frequency based on the data of magnetic flux density distribution contained in the PLOT file of the magnetic field analysis. It is possible to evaluate the iron loss by separating it into hysteresis loss and joule loss.

Furthermore, it allows the following analyses to be performed:

- An analysis to calculate loss from magnetic flux density and the direction components of X-axis, Y-axis, and Z-axis.
- An analysis to calculate loss from the radial and circumferential components of magnetic flux density that is performed using a coordinate system transformation.
- An analysis to calculate loss in a model including a rotational movement object.

MEMO It is possible to handle a 2D model and 3D model in iron loss calculation.

12-3-1-1. Methods of Loss Calculation

The iron loss is obtained with JMAG as follows.

$$Wloss = \sum_{i}^{nelem} (Wh_i(B, f) + We_i(B, f))$$

Wloss: Iron loss

Wh $_i$ (B): Hysteresis loss of each element

We $_i$ (B): Joule loss of each element

B: Magnetic flux density of each element

The following items can be selected for the formula above.

- Calculation formula to evaluate loss
  - How to use the experimental loss formula
  - How to use the loss data of the material data base
  - How to separate the loss into hysteresis loss and joule loss by referring to the iron loss file
- Method for calculating magnetic flux density used for iron loss calculation
  - FFT (frequency analysis)
  - Consider loop (selection is possible only in case of hysteresis loss)
  - Maximum value (selection is possible only in case of Joule loss)

The above are described below.

■ Calculation formula to evaluate loss

The calculation formula for evaluating loss can be chosen from one of the three methods described below. In all three calculation formulae, hysteresis loss and joule loss can be separately output. Furthermore, the calculation formula can be separately chosen for each material.

■ Using the experimental loss formula

The various coefficients of the formula for calculating loss can be set. Various coefficients used in the empiric formula can be specified separately.

■ Using the loss data of the material database

The iron loss is calculated from the loss density data of steel sheets for each frequency, which are provided as the material database. The user can prepare his/her own loss density data for each frequency (with User_Steel directory).

To separate hysteresis and joule losses, this technique adopts a frequency separation method. A frequency separation method is also used to complement the frequency at which data is not contained in the loss density data of the material database.

▶▶ Reference

For more information about the frequency separation method, see P.922 "12-3-12-13. Frequency Separation Method".

■ Separating the loss into hysteresis loss and joule loss by referring to the iron loss file

The loss is calculated by specifying the magnetic flux density data stored in the PLOT file obtained by magnetic field analysis, and both hysteresis loss density data and joule loss density data for each frequency. Hysteresis loss density data and joule loss density data must be prepared by the user.

Furthermore, in case a frequency not existing in the prepared loss density data is referred to during the calculation, it will be calculated by interpolation with the primary formula of the frequency at the time of hysteresis loss density, and with the secondary formula at the time of Joule loss density.

▶▶ Reference

For more information, see P.906 "12-3-12. Equations for Iron Loss Calculations".

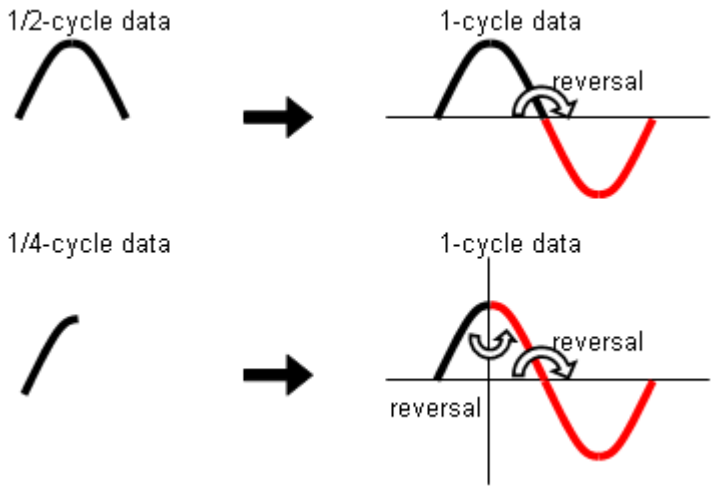
■ Methods for calculating magnetic flux density used for iron loss calculation

Select from the following techniques for calculating the magnetic flux density data used for iron loss calculation.

■ FFT (frequency analysis)

The amplitudes of magnetic flux density of each frequency component can be calculated by applying FFT frequency analysis to the time-series data of magnetic flux density. The resultant amplitudes are used.

The data of magnetic field analysis result of 1-cycle is required as the time series data of magnetic flux density. 1/2-cycle data and 1/4-cycle data can be used by extending it to 1-cycle data. However, this extension function of periodic character is done by reversal copying. Therefore, it can be used only for the model defined by a waveform that is symmetric with zero as it centers such as a sin curve.



Extending 1/2- or 1/4-cycle data

■ Consider loop (selection is possible only in case of hysteresis loss)

From the time series data of magnetic flux density, calculate the number and size of the hysteresis loops and obtain the iron loss. Similar to FFT (Frequency Analysis), result data equal to one cycle of the magnetic field analysis will be necessary.

■ Maximum value (selection is possible only in case of Joule loss)

Calculate the maximum value from the time series data of the magnetic flux density, and obtain the iron loss.

12-3-1-2. Iron Loss Analysis using Stress

The analysis using an iron loss that is increased because of compressive stress can be run (iron loss calculation).

- Specify a stress distribution when running an iron loss calculation using stress. The following setting methods for stress distribution are as follows.
  - Refer structural analysis results
  - Set stress value for each material
- Iron loss calculation using stress can only be used when material data with iron loss curve information for a stress dependency is used. The types of iron loss curve information for stress dependency are as follows.
  - Iron loss curve information included in the material data for a stress dependency created by user

▶▶ Reference

For more information about creating materials with iron loss curve information for a stress dependency, see P.884 "12-3-3-5. Iron Loss File (Stress-dependent)".

12-3-1-3. Types of Magnetic Field Analysis and Iron Loss Calculation

Depending on what type of analysis the referenced result file comes from, the contents of the iron loss calculation differs as follows.

■ Transient response magnetic field analysis (DP, TR)

When using the result of transient response magnetic field analysis in iron loss calculation, processing of the time series data needs to be conducted with either FFT, loop consideration, or maximum value consideration.

In addition, in order to define the interval of time series data to be used in iron loss calculation, the basic frequency needs to be set.

■ Frequency response analysis (FQ)

When using the result of frequency response magnetic field analysis in iron loss calculation, the frequency set in the frequency response analysis is used for the basic frequency. Although the outcomes of multiple steps will be displayed as the result, these correspond to the loss distribution at each step set in magnetic field analysis.

In addition, it is also possible to regard the outcomes of multiple steps as the result of frequency resolution and to output the sum of all steps.

■ Static analysis (ST)

When using the result of static magnetic field analysis in iron loss calculation, iron loss calculation can be conducted by setting the basic frequency.

The iron loss can be calculated for each of multiple steps set in static analysis.

12-3-2. Flow of Calculation (JMAG-Studio)

In this section, a step-by-step series of iron loss calculations are described as follows:

1. **Preparation of files to be referred to.**
   - PLOT file of magnetic field analysis
   - PLOT file of structural analysis
   - Coefficient and multiplier file
   - Iron loss file
   - Iron loss file (Stress-dependent)
   - Magnetic field analysis input file
2. **Start Calculate Iron Loss tool (basic configuration of the [Calculate Iron Loss] dialog box)**
3. **Set parameters of the [Basic] tab in the [Calculate Iron Loss] dialog box**
4. **Set parameters of the [Material] tab in the [Calculate Iron Loss] dialog box**
5. **Set parameters of the [Calculation] tab in the [Calculate Iron Loss] dialog box**
6. **Set parameters of the [Option] tab in the [Calculate Iron Loss] dialog box**
7. **Set parameters of the [User Subroutine] tab in the [Calculate Iron Loss] dialog box**
8. **Run iron loss calculation**
9. **Display results of iron loss calculation**

▶▶ Reference

For more information about the procedures from running the analysis to obtaining the iron loss calculation, see P.923 "12-3-13. Procedure for Evaluating the Loss".

12-3-3. Files Used for Iron Loss Calculation (JMAG-Studio)

In the iron loss calculation tool, there are cases in which the following files are used depending on the calculation method.

- PLOT file of magnetic field analysis
- PLOT file of structural analysis (In case stress distribution is put into consideration)
- Coefficient multiplier file (In case [Iron Loss Expression] is selected as the calculation method.)
- Iron loss file (In case [Use Loss File] is selected as the calculation method.)
- Iron loss file (Stress dependent)
- Magnetic field analysis input file for referencing rotation (Specify with [Option] tab in case the magnetic field analysis model of which loss is to be calculated contains a rotational motion.)
- Magnetic field analysis input file for the execution of iron loss calculation (in case analyses in the [Execute] dialog box is run from magnetic field analysis to iron loss calculation in succession)

  An explanation of these files will be given below:

12-3-3-1. Plot File of Magnetic Field Analysis

In the iron loss calculation, regardless of the differences in the calculation method, PLOT file of magnetic field analysis is necessary.

- These files may be exported from any magnetic field analysis modules among frequency response magnetic field analysis, transient response analysis, and static analysis.
- The plot file to use for the iron loss calculation needs to export the following information.
  - Magnetic flux density
  - Displacement

12-3-3-2. PLOT File of Structural Analysis

In case of conducting iron loss analysis that puts stress dependency into consideration, as a method of specifying the stress value, it is possible to select between a method in which a constant value is specified for each material, and a method in which the PLOT file of the structural analysis is referred to.

In case the PLOT file of the structural analysis is used, prepare this file prior to the iron loss analysis. Furthermore, when preparing the file, pay attention to the following points.

- Make certain that the results of the main stress are output to the PLOT file. The principal stress is the executive control condition of the structural analysis, and if the [Stress] check box is selected, it will be output.
- In the model of the PLOT file of the structural analysis referred to, make certain that the model of the iron loss calculation is included. Even if it is necessary for the iron loss calculation to refer to results of the structural analysis, in case the stress value of the corresponding space is not found, iron loss calculation which puts stress into consideration cannot be conducted.

  (Examples)
  - Structural analysis referred to is 1/2 model, iron loss calculation is 1/2 divided model.=>=> Analysis is possible.
  - Structural analysis referred to is full model, iron loss calculation is 1/2 divided model. =>=> Analysis is possible.
  - Structural analysis referred to is 1/4 divided model, iron loss calculation is 1/2 divided model. =>=> Analysis is impossible.
- It does not matter even if the shapes of the mesh are different between the model of the PLOT file of structural analysis and the model of the iron loss calculation.

12-3-3-3. Coefficient/Multiplier File

The coefficient/multiplier file (*.txt) is used in case iron loss type is selected as the material type of the iron loss calculation.

By using this file, it is possible to specify the coefficients *Kh*, *Ke*, and Multipliers α, β, γ, and δ in the following Iron Loss equation.

$$W = Kh \cdot B^{\alpha} \cdot f^{\beta} + Ke \cdot B^{r} \cdot f^{\delta}$$

W: Loss density (W/m^3)

B: Magnetic flux density (T)

f: Frequency (Hz)

In case of preparing the coefficient/multiplier File, pay attention to the following points.

- The coefficient/multiplier file is in the form of text (*.txt). Conduct editing by utilizing text editor, etc.
- As shown in the example below, create a file in matrix form using an array of magnetic flux density (T) in the row header and an array of frequency (Hz) in the column header.
  Example)

```
                50          100         200
0.5          K11         K12         K13
1            K21         K22         K23      < Frequency
1.5          K31         K32         K33
2            K41         K42         K43

^ Magnetic flux density
```

- Spaces, tabs, and commas can be used to delimit items.
- Set magnetic flux density and frequency in either ascending or descending order.
- Coefficients related to frequency and magnetic flux density given by a file can be obtained by using a linear interpolation scheme.
- When magnetic flux density and frequency exceed the range given by the file, the coefficients will be set to zero. When it is set to zero, the element ID and frequency (when the loss calculation method is FFT) will be output to a warning file (*.warning).

12-3-3-4. Iron Loss File

In case [Use Loss File] is selected as the calculation method, specify the iron loss files (*.iron) that refer to the hysteresis loss and joule loss, respectively.

The points to be careful of in case the iron loss files are created arbitrarily are explained below.

■ File format

Card 1

Title (up to 80 characters)

Card 2

Free (up to 80 characters)

Card 3

1-10 Number of frequencies

Card 4

1-10 Order number

(repeat up to the number of frequencies)

11-30 Frequency

Card 5

1-10 Number of point sequences

Card 6

1-20 Magnetic flux density (T)

21-40 Loss density (W/m^3)

(repeat up to the number of point sequences)

■ Sample

```
sample                                                                  Card 1
                                                                        Card 2
                2                                                       Card 3
                1       5.000000000000e+01                              Card 4
                3                                                       Card 5
      0.000000000000e+00        0.000000000000e-00                    Card 6
      5.000000000000e-01        0.500000000000e+03
      1.000000000000e+00        0.500000000000e+05
                2       1.000000000000e+02                              Card 4
                3                                                       Card 5
      0.000000000000e+00        0.000000000000e-00                    Card 6
      5.000000000000e-01        0.500000000000e+03
      1.000000000000e+00        1.500000000000e+05
```

- Enter data right-justified in relation to the number of rows set in the card.
- Enter integers in 10 digits and real numbers in 20 digits.
- File format is shown in the following. Enter integers in 10digits and real numbers in 20digits in right justification.
- Create the above-mentioned format file with a file name of extension iron, by using a text editor, etc. Place the created file in the following directory.

  `(JMAG installation directory)\database\user_steel`

- Do not include spaces or colons (:) in the file name.

12-3-3-5. Iron Loss File (Stress-dependent)

The stress-dependent iron loss properties can be defined in the iron loss file.

- To create a iron loss file for stress-dependent loss properties, use the following format and an extension *.csv.
  - The data should include the iron loss properties for each stress value.
  - Create data so that the stress and frequency values are in ascending order.

■ File format

Line	Details
1	Data title
2	"1"
3	Number of stress values
4	Stress (Pa), Number of frequencies: repeat as many times as the # of stress values.
5	Frequency (Hz), number of flux density vs. loss density point sequence data: repeat as many times as the # of frequencies.
6	Flux density (T), loss density (W/m^3) repeat as many times as the # of point sequence data.

MEMO Specify the iron loss characteristics for each stress value.

MEMO Do not change the second line.

MEMO Specify the stress value and frequency in ascending order.

■ Example

```
Sample for Stress Dependent Ironloss
1
2
-2.0e+7, 4
50, 9
0,0
0.2,0.005
0.4,0.22
0.6,0.43
and so on.
```

- The stress-dependent loss file is used for the setup of materials in the magnetic field analysis.

1. **Create the iron loss file.**
2. **Open the [Material] dialog box in the magnetic field analysis, and click the [Setting] button on the [Magnetizing Properties] tab.**
3. **In the [Magnetizing Properties] dialog box, select "User_Steel (Material Database)" from the [Type] combo box.**
4. **Click the [Setting] button in the [Point Sequence] group box.**
5. **In the [Material Database] dialog box, select the [Magnetizing Properties (stress dependent)] radio button under [Data] Type.**
6. **Select the [Show Iron Loss] check box.**
7. **To add the created data to the material database, click the [Entry] button.**

   The [Open] dialog box is shown.
8. **Select the CSV file created in step i.**

   The [Material Database] dialog box is shown.

   The created iron loss property is added to the list. If a property with the same already exists in the material database, a confirmation message appears, asking whether to overwrite or not.

   The created iron loss property data is saved to the directory specified for [User Material DB Directory] on the [Save] tab in the [Preference] dialog box opened from the [Options] menu.

12-3-3-6. Magnetic Field Analysis Input File for Referencing Rotation

In case the magnetic field analysis model of which the loss is to be calculated makes a rotational movement, specify either the JCF file referred to with the [Option] tab of the [Calculate Iron Loss] dialog box or the solver data file.

At this time, it is necessary for the JCF file or the Solver data file that can be specified to satisfy the following points.

- The settings of the materials and the conditions are completed.
- Mesh is created.

MEMO In case calculation is made by using patch mesh function or adaptive mesh function in the region data or solid data for version 8.2 or earlier, mesh information is not included in the JCF file or solid file that are created. Therefore, such files could not be used for iron loss calculations.
In case the same calculation is executed with version 8.3 or later, a file called "*.input.jcf" will be output after the calculation is completed. This file is made by inputting the mesh information to the JCF file that executed the calculation. It is possible to specify this "*.input.jcf" as magnetic field analysis input file.
In the "" of the "*.input.jcf" the name of the JCF file that executed the calculation will be entered.

12-3-3-7. Magnetic Field Analysis Input File for Iron Loss Calculation (*.jcf)

After setting with the iron loss tool, it is possible to export to the JCF file by selecting [File] > [Save as] from the menu bar of the [Calculate Iron Loss] dialog box. This will enable the successive performance of iron loss calculation from the [Execute] dialog box.

12-3-4. Starting/Exiting the Calculate Iron Loss Tool (JMAG-Studio)

The settings for the iron loss calculation can be specified in the [Calculate Iron Loss] dialog box.

12-3-4-1. Starting the Calculate Iron Loss Tool

- **Select [Tools] > [Calculate Iron Loss] from the menu bar in JMAG-Studio.**

- **Select [Iron Loss] in the [Condition] dialog box. The iron loss condition (226 page) can be set for the magnetic field analysis in JMAG-Studio.**

If the iron loss condition is set for a magnetic field analysis, the analysis is run for the project that is being open, and then the iron loss calculation is performed using the results of the project.

12-3-4-2. Exiting the Calculate Iron Loss Tool

1. **Select [File] > [Exit] from the menu bar in the [Calculate Iron Loss] dialog box.**

12-3-4-3. Menus in the [Calculate Iron Loss] Dialog Box

- [File]
  - [New]: Clears data set in the dialog box and returns to the initial state.
  - [Open]: Opens a file. The following files can be imported:
    - JCF file (*.jcf)
    - Iron loss parameter file * Any extension can be used.
  - [Save]
  - [Save as]: Saves a file. The following files can be exported:
    - JCF file (*.jcf): The input file for the iron loss calculation is exported here.
    - Iron loss parameter file * Any extension can be used.
  - [Exit]: Exits the Calculate Iron Loss tool.
- [Tools]
  - [Merge Loss Files]: Combines the loss file of magnetic field analysis and the loss file of the Calculate Iron Loss tool into one loss file (*.loss) that is used for a thermal analysis.

▶▶ Reference
For more information, see P.888 "12-3-4-5. Merging Loss Files".

- [Help]
  - [About]: displays the version of the iron loss calculation module (LS).

MEMO The menus in the [Calculate Iron Loss] dialog box are available when the Calculate Iron Loss tool is started from the menu bar. They are not available when the iron loss condition is set.

12-3-4-4. Tabs in the [Calculate Iron Loss] Dialog Box

This dialog box consists of four tabs.

- [Basic] tab
- [Material] tab
- [Calculation] tab
- [Option] tab
- [User Subroutine] tab

▶▶ Reference

For more information, see the descriptions included in a later section.

12-3-4-5. Merging Loss Files

A loss file (*.loss) used in a thermal analysis can be created by merging the loss file (*.loss) of a magnetic field analysis and the loss file (*.loss) of an iron loss analysis into one file.

1. Select [Tools] > [Calculate Iron Loss] from the menu bar.

The [Calculate Iron Loss] dialog box is displayed.

2. Select [Tools] > [Merge Loss Files] from the menu bar.

The [Merge Loss File] dialog box is displayed.

3. Specify the loss file of a magnetic field analysis to merge.

Click [...] next to the [Loss file for magnetic field analysis] box, and then specify the file.

4. Specify the loss file of an iron loss analysis to merge.

Click [...] next to the [Loss file for iron loss calculation] box, and then specify the file.

5. Enter a name for the loss file for a thermal analysis in the [Loss file for thermal analysis] box.

6. Click [OK].

A loss file is created. This file can be specified for the heat source condition of a thermal analysis.

■ Loss file format

The format for loss file (*.loss) are the following.

Line 1: Analysis time [sec], lines after line 2

Line 2: Element ID, loss density [W/m3]

MEMO Integers can be exported in 10 digits, and real numbers in 20 digits.

■ Sample

The following example describes the loss density for eight components at 1 millisecond.

The element ID 1 thorough 4 is the loss density for 1e-10 [W/m3].

The element ID 5 thorough 8 is the loss density for 2e-10 [W/m3].

1.0000000000E-03	8
1	1.0000000000E-10
2	1.0000000000E-10
3	1.0000000000E-10
4	1.0000000000E-10
5	2.0000000000E-10
6	2.0000000000E-10
7	2.0000000000E-10
8	2.0000000000E-10

Example of loss file

12-3-5. Settings in the [Basic] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)

1. Specify the following items.

- [Reference Plot File]: Specify the plot file obtained from a magnetic field analysis. The iron loss is calculated using the magnetic flux density distribution contained in this file.

  Click [...] to open the [Open] dialog box. Select the file from the list or enter the file name in the text box, and then click [OK].

MEMO If a material file (e.g., data.smt) and a plot file (e.g., data.plot) are saved in the same folder, the materials in the material file are automatically added to the material list in the plot file. The material file is exported from JMAG-Studio 6.0 or later.

- [Basic Frequency]: Specifies the frequency for one period of magnetic flux density.

  The basic frequency can be specified in one of the following ways:

  - [Calculate from Poles and Revolution Speed]: Calculates the basic frequency based on the number of poles and the rotational angle. This setting is used for rotating machineries.
    - [Poles]
    - [Revolution Speed (rpm)]

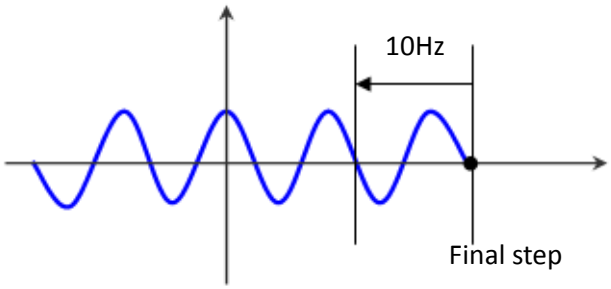
MEMO The basic frequency is calculated as follows:

$$f = \frac{n}{60} \times \frac{P}{2}$$

f: Basic frequency (Hz), *P*: Number of poles, *n*: Rotation speed (rpm)

- [Set Frequency (Hz)]: Specify the frequency in the text box. This setting is used for models other than rotating machineries.

MEMO The range in the time series data of the magnetic flux density that is used for the iron loss calculation is determined as follows.
For example, if a basic frequency is 10 Hz and if [Reference Start Step] and [Reference End Step] in the [Calculation] tab are not set, the range from the final step of the time series to 10 Hz is used for the iron loss calculation.



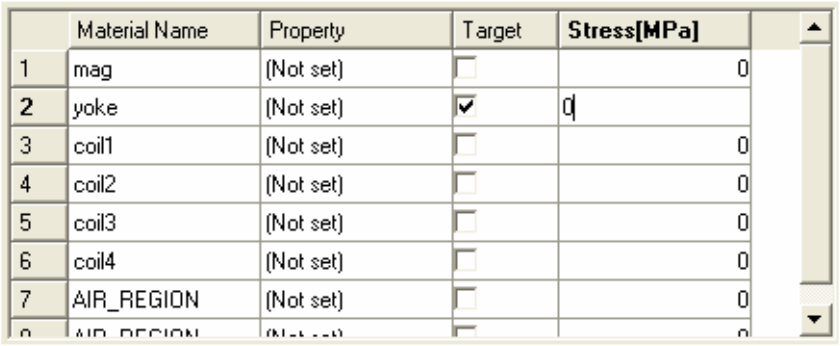
- [Output File]: Specify the folder to save the result file of the iron loss calculation. Click [...] to open the [Save As] dialog box. Specify the folder, and then enter a file name.

12-3-6. Settings in the [Material] Tab in the [Calculate Iron Loss] Dialog Box (JMAG-Studio)

In the [Material] tab, the target and method for the loss calculation can be set. The iron loss is calculated for each material.

1. Specify each item for each material.

The following table is displayed in the [Material] tab.



	Material Name	Property	Target	Stress[MPa]
1	mag	(Not set)	☐	0
2	yoke	(Not set)	☑	0
3	coil1	(Not set)	☐	0
4	coil2	(Not set)	☐	0
5	coil3	(Not set)	☐	0
6	coil4	(Not set)	☐	0
7	AIR_REGION	(Not set)	☐	0

Each item in the table is as follows.

- [Material Name], [Property]: Displays the material name specified by the user and material type. These settings can be modified by clicking the [Detail] button next to the table.
- [Target]: Specify whether or not to include the material in the loss calculation. If the material in the material database is used in the JCF file (*.jcf) of the magnetic field analysis, the check box is selected by default.
- [Stress (MPa)]: Specify the stress value for each material to take into account the stress in the loss calculation. However, if [Set Each Material] is not selected under [Stress Distribution], the specified values are not used in the loss calculation.

MEMO In the case of the iron loss condition, even if a material in the list is selected, the corresponding area in the model will not be highlighted.

2. Select a material from the list, and then click [Detail] for further settings.

The [Material Details] dialog box is displayed.
The following items can be specified:

- [Material Name]
- [Material Type]: Select the method to define the loss from the following three types.
  - [Material database]: Defines the loss by using the loss data in the material database.
  - [Iron loss expression]: Defines the loss by using the empirical formula of loss.
  - [Use loss file]: Defines the loss by referencing the iron loss file (*.iron). The iron loss file can be created.

    ▶▶ Reference

    For information about the format for the iron loss file, see P.879 "12-3-3. Files Used for Iron Loss Calculation (JMAG-Studio)".
- [Referenced Components of Magnetic Flux Density]: Select the check boxes of the components of magnetic flux density to use for the iron loss calculation.
- [Calculation Type]
  - [Absolute]: Calculates the absolute value of magnetic flux density from the selected components, and then calculates the loss using this value. Select this if the alternating magnetic field is dominating.
  - [Each Component]: Calculates the magnetic flux density for each component, and then calculates the loss using the obtained value. Select this if the rotating magnetic field is dominating.
    To apply the same setting (material database, iron loss expression, or use loss file) to all components, select the [Use Same Setting] check box.
- [1st Component], [2nd Component], [3rd component]

  Specify the loss for each component. The settings vary depending on the method selected from the [Material Type] list.
  - If [Material database] is selected from the [Material Type] list: Click [Select] to open the [Material Database] dialog box. Select the manufacturer and the type of material, and then select the material to use from the list.
    Click [OK] to return to the [Material Details] dialog box.

- If [Iron loss expression] is selected from the [Material Type] list: The following equation is used to calculate the iron loss.

$$W = Kh \cdot B^{\alpha} \cdot f^{\beta} + Ke \cdot B^{r} \cdot f^{\delta}$$

W: Loss density (W/m^3), *B*: Magnetic flux density (T), *f*: Frequency (Hz)

Specify coefficients *Kh* and *Ke*, and multipliers α, β, γ, and δ.
Click [Set] to open the [Coefficient Table] dialog box.
Click [Import from jcf] to use the coefficient table that is already created, and then specify the JCF file. Only the coefficient table data contained in the specified JCF file is read.

MEMO The JCF file specified here is the file saved by selecting [File] > [Save As] in the Calculate Iron Loss tool.

Click [Add] to specify coefficient and multiplier.
Click [Modify] to modify the coefficient table. The [Coefficient Setting] dialog box is displayed. Specify a name in the [Expression Name] box. Select either [Constant] or [By File] for each coefficient and multiplier. If [Constant] is selected, enter the value in the text box. If [By File] is selected, click [...], and specify the coefficient file (*.txt).

Reference
For information about the format for coefficient file, see P.879 "12-3-3. Files Used for Iron Loss Calculation (JMAG-Studio)".

After the settings are complete, click [OK] to return to the [Coefficient Table] dialog box. Select the data to use from the list, and then click [OK].

- If [Use loss file] is selected from the [Material Type] list: The iron loss file (*.iron) in the following directory is displayed for each list.

```
(JMAG installation folder)\Database\User_Steel
```

Or,

```
User material DB directory
```

MEMO The user material DB directory can be specified in the [Save] tab of the [Preferences] dialog box. To display the [Preferences] dialog box, select [Options] > [Preferences] from the menu bar in JMAG-Studio.

Specify the loss files to reference for hysteresis loss and Joule loss.

Reference
For information about the format for iron loss file, see P.879 "12-3-3. Files Used for Iron Loss Calculation (JMAG-Studio)".

- [Coordinate System Conversion]: Specify the coordinate system conversion feature to obtain the magnetic flux density for each component. Select from the following three types.
  - [Not Use]: Do not use the coordinate system conversion feature. In this case, the global coordinate system is used.

    Each component is specified as follows.

    1st component = X-component

    2nd component = Y-component

    3rd component = Z-component
  - [Rectangular]: Converts the magnetic flux density vector to the rectangular coordinate system, and then obtains the magnetic flux density.

    In this case, each component is converted as follows.

    1st component = X-component of the coordinate system after conversion

    2nd component = Y-component of the coordinate system after conversion

    3rd component = Z-component of the coordinate system after conversion

    Click [Set] to open the [Coordinates System Table] dialog box.
    Click [Add] to create a new coordinates system
    Click [Modify] to modify the existing coordinate system. The [Coordinates System] dialog box is displayed. Specify the following items.
    - [Origin]: Specify the origin of the rectangular coordinate system.
    - [Point on X-axis]: Specify a point on the X-axis in the rectangular coordinate system.
    - [Point on XY-plane]: Specify a point on the XY-plane in the rectangular coordinate system.

    For example, when specifying a rectangular coordinate system for which the origin is (1, 0, 0), and the X-, Y-, and Z-axis are parallel to the X-, Y-, and Z-axis of the global coordinate system.

    Origin = (1, 0, 0)
    Point on X-axis = (2, 0, 0)
    Point on XY-plane = (2, 1, 0)

    After specifying each item, click [OK] to return to the [Coordinates System Table] dialog box. Select the coordinate system to use from the list, and then click [OK].

- [Cylindrical]: Converts the magnetic flux density vector to the cylindrical coordinate system, and then obtains the magnetic flux density.

  In this case, each component is converted as follows.

  1st component = r-component

  2nd component = theta-component

  3rd component = z-component

  For example, the loss is calculated using the r- and theta-component of the magnetic flux density in the global coordinate system if the [1st Component] and [2nd Component] check boxes are selected for [Referenced Components of Magnetic Flux Density].

  Click [Set] to open the [Coordinate System Table] dialog box.
  Click [Add] to create a new coordinate system.
  Click [Modify] to modify the existing coordinate system. The [Coordinates System] dialog box is displayed. Specify the following items.

  - [Origin]: Specify the origin of the cylindrical coordinate system.
  - [Point on X-axis]: Specify a point on the X-axis in the cylindrical coordinate system.
  - [Point on XY-plane]: Specify a point on the XY-plane in the cylindrical coordinate system.

    For example, when specifying a cylindrical coordinate system for which the origin is (1, 0, 0) and the axis of the cylinder (Z-axis) is parallel to the Z-axis of the global coordinate system.

    Origin = (1, 0, 0)
    Point on X-axis = (2, 0, 0)
    Point on XY-plane = (2, 1, 0)

  After specifying each item, click [OK] to return to the [Coordinates System] dialog box. Select the coordinate system to use from the list, and then click [OK].

After the settings are complete, click [OK] in the [Material Details] dialog box to return to the [Calculate Iron Loss] dialog box.

3. Select the method to define the stress distribution, and then specify the necessary Items.

- [Not Apply]: Ignores the stress values specified for each material in the list during the loss calculation.
- [Apply Stress Analysis Result]: Defines the stress values by referencing the plot file of a structure analysis. Click [...] to open the [Open] dialog box, and then specify the plot file of a structure analysis to use.

  ▶▶ Reference

  For information about the plot file of a structural analysis, see P.879 "12-3-3. Files Used for Iron Loss Calculation (JMAG-Studio)".
- [Set Each Material]: Uses the stress value specified for each material in the table during the loss calculation.

  The stress-dependent materials are provided in the material database of Nippon Steel Corporation.

  Enter a negative value (compressive stress value) in the [Stress (MPa)] column when specifying a stress value for each material.

  Example: -10 MPa

12-3-7. Settings in the [Calculation] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)

1. Select the method to extract the magnetic flux density when obtaining the hysteresis loss.

Select from the following two types.

- [Apply Loop]: Calculates the maximum value of magnetic flux density from the time series data of the magnetic flux density while taking into account the hysteresis loop. The maximum value is used for the loss calculation.
- [FFT]: Transforms the time series data of magnetic flux density using the Fast Fourier Transform (FFT), and calculates the amplitude of the magnetic flux density for each frequency component. The amplitudes are used for the loss calculation.

2. Select the method to extract the magnetic flux density when obtaining the joule loss.

Select from the following two types.

- [Max Value]: Calculates the maximum value of magnetic flux density from the time series data of the magnetic flux density. The maximum value is used for the loss calculation.
- [FFT]: Transforms the time series data of magnetic flux density using the Fast Fourier Transform (FFT), and calculates the amplitude of the magnetic flux density for each frequency component. The amplitudes are used for the loss calculation.

3. If [FFT] is selected in either step 1 or step 2, specify the harmonic order that is to be exported to the plot file of the iron loss calculation.

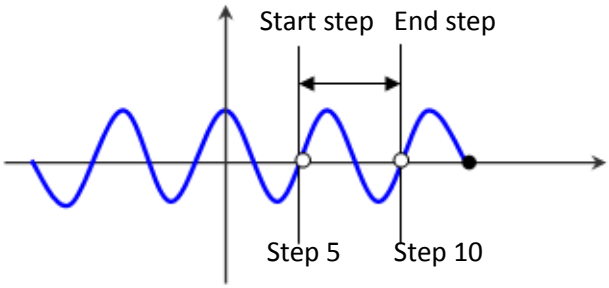
- [All]: Exports the loss for each order and the total loss for all of the orders.
- [Specify]: Exports the loss for each specified order and the total loss of the specified orders. Enter the order in the [Harmonic Order] box.

  MEMO When specifying multiple orders in the [Harmonic Order] box, use the comma such as '1, 5, 9' or use the hyphen such as '1-10'.

4. Specify the reference start step and reference end step to define the range in the time series data of the magnetic flux density using steps.

- [Reference Start Step] / [Reference End Step]: Specify the range in the time series data used for the loss calculation. Select each check box, and enter a value in each text box.

  Example: The start and end steps are '5' and '10', respectively.



The range can also be specified by combining the basic frequency specified in the [Basic] tab and [Reference Start Step] or [Reference End Step].

- Specify [Basic Frequency] and [Reference Start Step]: Specify the range for one basic frequency from the start step.
- Specify [Basic Frequency] and [Reference End Step]: Specify the range for one basic frequency from the end step.

MEMO When the basic frequency, start step, and end step are all specified, the range specified by the start step and end step is used for the loss calculation.

5. If multiple steps are specified in step 4, select the [Total Iron Loss of All Frequencies (FQ)] check box to obtain the sum total.

The sum total of multiple steps is indicated at step 0.

12-3-8. Settings in the [Option] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)

1. Specify each item as follows.

- Calculating the loss for different frequencies: The loss for different frequencies can be calculated by scaling the time axis of waveform of the magnetic flux density based on the assumption that the waveform of the magnetic flux density does not vary with the increase of the revolution speed of a rotating machinery.

  Enter a multiplier for the basic frequency in the [Frequency Scale Factor] box.

  With this setting, the loss is re-calculated for the frequency that is obtained by multiplying the frequency specified in the [Basic Frequency] box of the [Basic] tab.

  For example, when [Basic Frequency] is set to 20 Hz and [Frequency Scale Factor] is set to 2, the loss is calculated for 40 Hz after the loss is calculated for the basic frequency. The results of each calculation are saved in different plot files.

  More than one multiplier can be specified. There is no limit to the number of multipliers to specify.

  If not specified, only the results of a calculation for the basic frequency are exported.

  MEMO If multiple frequencies are specified, each plot file is named using the file name and multiplier.
  Example: If the name of the plot file specified is "loss.plot" and the multipliers are "2" and "3," the results are exported to three files named "loss.plot," "loss2.plot," and "loss3.plot."

- Calculating the loss for a magnetic field analysis model that includes rotating parts: Select [On] for [Rotation Motion].
  The path to the JCF file (or solver data file) is displayed in the [Reference solver file] box. This JCF file has the same name as the plot file specified in [Reference Plot File] of the [Basic] tab, and is saved in the same folder as the plot file.

  With this setting, the Calculate Iron loss tool reads the rotational motion settings from the specified JCF file (or solver data file), converts the magnetic flux density vector of the rotating model into a value in the rotational motion system, and then calculates the loss.

  To specify another JCF file (or solver data file), click [...].

  To use the file containing the results using the adaptive mesh option or patch mesh option, click [...], and then specify a file named "*_input.jcf" (* indicates the analysis name).

MEMO If this setting is not made for a model that includes rotating parts, the loss will not be correctly calculated in a orthogonal coordinate system. However, the loss will be correctly calculated by setting a cylindrical coordinate system about the rotation axis in the detailed settings of the [Material] tab.

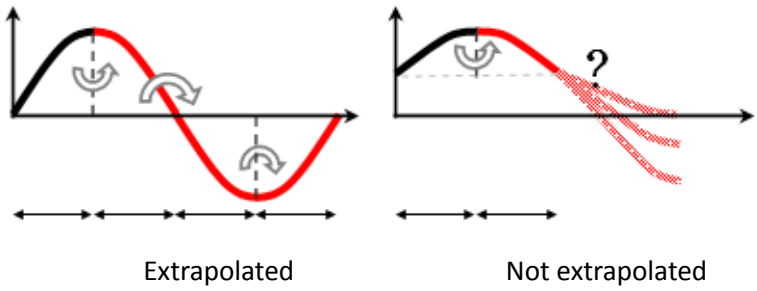
▶▶ Reference
For information about the JCF file/solver file to reference, see P.879 "12-3-3. Files Used for Iron Loss Calculation (JMAG-Studio)".

- Calculating the loss using the periodicity of the time-series data: Extrapolates the time series data based on the symmetry of the waveform of magnetic flux density when the waveform is shorter than one period.

  Select any one of the following.

  - [None]: Do not use the periodicity of the time series data. This is selected by default.
  - [1/2]: Extrapolates the time series data to one period when the range specified by [Reference Start Step] and [Reference End Step] in the [Calculation] tab is regarded as a 1/2 period.
  - [1/4]: Extrapolates the time series data to one period when the range specified by [Reference Start Step] and [Reference End Step] in the [Calculation] tab is regarded as a 1/4 period.

    However, the 1/4 period data can be extrapolated only when the data starts from phase 0.



Example of the 1/4 period data

MEMO To use this feature, be sure to specify the start step and end step in the [Calculation] tab.

- Dividing the plot file of an iron loss analysis into multiple files by specifying the file size: Enter the file size in the [Division of File (MByte)] box. With this setting, the data is saved in a new file each time the file reaches the specified size during the loss calculation.

12-3-9. Settings in the [User Subroutine] Tab of the [Calculate Iron Loss] Dialog Box (JMAG-Studio)

1. Specify the following items when using a user subroutine.

- [Select User Subroutine DLL] : Select a user subroutine DLL to use for the analysis. The user subroutine DLL files located in the following directory are displayed in the list.

  `\JMAG-Studio installation folder\usrsub\ironloss`

  If using a user subroutine DLL file on a remote machine when running an analysis on the remote machine, the user subroutine DLL file needs to be added to the client machine. Click [Add], and then specify the following items in the [User Subroutine DLL Name] dialog box.
  - [User Subroutine DLL Name]: Enter the name of the user subroutine DLL file. The extension is not required.
  - [Title]: Enter a title for the compiled user subroutine DLL. This is not used for the analysis.
  - [Description]: Enter a brief description for the compiled user subroutine DLL. This is not used for the analysis.

  When the settings are complete, click [OK] to return to the [User Subroutine] tab. Select the added file from the [User Subroutine DLL Name] list.
  Click [Modify] to modify the settings.

  Save the user subroutine DLL (shared library) on the remote machine in the following directory.

  - Windows

    `\JMAG installation folder\usrsub\ironloss`

  - Linux

    `/works/solver/mod/usrsub/ironloss/`

  MEMO If distributing the calculation that uses the user subroutine DLL, place the user subroutine DLL (for Windows machine) or shared library (for Linux machine) on all of the CPU nodes in the CPU group to use for the distributed calculation.

  ▶▶ Reference
  - For information about creating a user subroutine, see "3-56. User Subroutine" on page 330.
  - For more information about the arguments for iron_str, see P.927 "12-3-14. User Subroutine iron_str".

12-3-10. Running Iron Loss Calculation (JMAG-Studio)

12-3-10-1. Running Iron Loss Calculation

1. **Click [Run] in the [Calculate Iron Loss] dialog box.**

   While running a calculation, the [Terminated] dialog box appears. Reports on the progress of a calculation are shown in the [Terminated] dialog box.

   If there are any errors in the settings, the [Error] dialog box will be displayed. Correct the parameters causing errors as indicated by the dialog message.

   MEMO In case magnetic field analysis project is set as the iron loss condition without using the iron loss calculation tool, after executing the calculation of that project, the iron loss calculation will be executed automatically.

12-3-10-2. Running Multiple Iron Loss Calculations

1. **Set the parameters for each tab in the [Calculate Iron Loss] dialog box.**
2. **Export the input file (*.jcf) for iron loss calculation through [File] > [Save As] in the [Calculate Iron Loss] dialog box.**
3. **Repeat Steps 1 and 2, to create a number of input files equal to the number of successive running.**
4. **Start the [Execute] dialog box.**
5. **Enter the data prepared in Step 3 in [Job List] and click [Run].**

12-3-10-3. Operations in the [Running Calculation] Dialog Box

1. **The calculation will be interrupted if [Cancel] in the [Running Calculation] dialog box is clicked.**

   When the calculation is completed or interrupted, the [Cancel] button will change to the [Close] button.

12-3-11. Displaying Results (Output Items of Iron Loss Analysis)

[Calculation of Iron loss normal end] is displayed in the [Running Calculation] dialog box when the calculation is run normally. A plot file for the iron loss calculation is exported with the name specified in the [Output File] box of the [Calculate Iron Loss] dialog box.

The following items will be output to the result file.

Output item	Unit	Type	Object	Note
Iron loss density	W/m^3	scalar	element	
Hysteresis loss density	W/m^3	scalar	element	
Joule loss density	W/m^3	scalar	element	
Iron loss	W	step	element	
Hysteresis loss	W	step	-	
Joule loss	W	step	-	
Iron loss density (each component)	W/m^3	scalar	element	In case [Each Component] is selected in the [Calculation Type] of the [Material Details] dialog box, this is output.
Hysteresis loss density (each component)	W/m^3	scalar	element	In case [Each Component] is selected in the [Calculation Type] of the [Material Details] dialog box, this is output.
Joule loss density (each component)	W/m^3	scalar	element	In case [Each Component] is selected in the [Calculation Type] of the [Material Details] dialog box, this is output.
Iron loss (each component)	W	step	element	In case [Each Component] is selected in the [Calculation Type] of the [Material Details] dialog box, this is output.
Hysteresis loss (each component)	W	step	-	In case [Each Component] is selected in the [Calculation Type] of the [Material Details] dialog box, this is output.
Joule loss (each component)	W	step	-	In case [Each Component] is selected in the [Calculation Type] of the [Material Details] dialog box, this is output.

Output item	Unit	Type	Object	Note
Reference flux density	T	vector	-	This is output when [FFT] is selected.
Reference flux density (component)	T	scalar	-	This is output when [FFT] and [Each Component] in [Calculation Type] are selected.

MEMO Each of the export items are displayed in units that are applied to an analysis model in JMAG-Designer.

MEMO Joule loss is equal to eddy current loss, and Joule loss density is equal to eddy current loss density.

Pay attention to the following points regarding the output items.

- In Step 1, the total of Iron loss will be output.
- In case calculation is made for each component, the corresponding results will be output for each component of the magnetic flux density.
- In case the [FFT] radio button is selected with the [Calculation] tab for either the hysteresis loss or Joule loss, the magnetic flux density distribution for each frequency will be output in complex number values. In case the [Apply Loop] radio button is selected in the hysteresis loss, and in addition, if the [Max value] radio button is selected in the Joule loss, the magnetic flux density distribution for each frequency will not be output.
- The iron loss, hysteresis loss, and Joule loss are converted during the calculation according to the conversion factor written in the result file of the magnetic field analysis.

MEMO Flux density and reference flux density

The flux density is output as a result of a magnetic field analysis in which the magnets and coil of the analysis model produce a magnetic field. Use the flux density data to check the strength and saturation of the magnetic field.

The reference flux density is a component of flux density that contributes to the iron loss and is output as a result of iron loss analysis. The reference flux density is extracted from the time varying flux density obtained from the magnetic field analysis. The iron loss calculation tool calculates the iron loss density from the iron loss curve based on the reference flux density and frequency. To verify iron loss density, use the reference flux density.

12-3-12. Equations for Iron Loss Calculations

Equations for use in evaluating the loss are described. Equations to be applied differ depending on the combination of the following conditions.

■ Selection 1; Type of equation

- Using experimental loss formula.
- Using loss data of material database.
- A method in which the loss file is referred to, and specification is made separately for hysteresis loss and Joule loss.

■ Selection 2; Type of magnetic flux density to be used

- FFT (frequency distribution)
- Maximum value * In case of joule loss.
- Apply loop * In case of hysteresis loss.

■ Selection 3; Type of loss to be calculated

- Hysteresis loss
- Joule loss

Note that the following sections present equations for evaluating the loss for each combination of the above selections.

12-3-12-1. Equation A - 1

- Type of equation: This method uses experimental loss formula
- Type of magnetic flux density to be used: FFT (frequency analysis)
- Type of loss to be determined: Hysteresis loss

The equation used to evaluate the loss is as follows.

Equation A-1: $$Wh[W] = \sum_{e=1}^{nelem} \left\{ \sum_{k=1}^{N} (Kh \times |B_k|^{\alpha} \times f_k^{\beta}) \times V_e \right\}$$

- Magnetic flux density $|B_k|$ (T) at the frequency order k:
  - Evaluating the loss for each component of magnetic flux density:

    $$|B_k| = \sqrt{B_k * B_k^*}$$

    B_k: Value (complex number) after FFT for each component "X, Y, Z, or r, theta, z" of magnetic flux density

    B_k^*: Complex conjugate numbers of B_k

  - Determining the loss from the amplitude of the magnetic flux density:

    $$|B_k| = \sqrt{B1_k * B1_k^* + B2_k * B2_k^* + B3_k * B3_k^*}$$

    $B1_k$, $B2_k$, $B3_k$: Value (complex number) after FFT for each component "X, Y, Z or r, q, z" of magnetic flux density

    $B1_k^*$, $B2_k^*$, $B3_k^*$: Complex conjugate numbers of $B1_k$, $B2_k$, $B3_k$

- Coefficients of hysteresis loss Kh, Kh= constant, or $Kh(|B_k|, f_k)$
- Frequency at the frequency order k: f_k (Hz): $f_1, f_2, f_3, \ldots f_N$
- Maximum frequency order: N
- Multipliers/coefficients of the empiric formula: alpha, beta
- Number of elements: nelem

12-3-12-2. Equation A - 2

- Type of equation: Using experimental loss formula.
- Type of magnetic flux density to be used: FFT (frequency analysis)
- Type of loss to be determined: Joule loss

  The equation used to determine the loss is as follows.

Equation A-2: $$We[W] = \sum_{e=1}^{nelem} \left\{ \sum_{k=1}^{N} (Ke \times |B_k|^{\gamma} \times f_k^{\delta}) \times V_e \right\}$$

- Magnetic flux density $|B_k|$ (T) at frequency order k:
  - Determining the loss for each component of magnetic flux density:

    $$|B_k| = \sqrt{B_k * B_k^*}$$

    B_k: Value "complex number" after FFT for each component "X, Y, Z or r, theta, z" of magnetic flux density

    B_k^*: Complex conjugate numbers of B_k

  - Determining the loss from the amplitude of the magnetic flux density:

    $$|B_k| = \sqrt{B1_k * B1_k^* + B2_k * B2_k^* + B3_k * B3_k^*}$$

    $B1_k$, $B2_k$, $B3_k$: Value (complex number) after FFT for each component (X, Y, Z or r, theta, z) of the magnetic flux density

    $B1_k^*$, $B2_k^*$, $B3_k^*$: Complex conjugate numbers of $B1_k$, $B2_k$, $B3_k$

- Coefficients of Joule loss *Ke*, *Ke*= constant, or $Ke(|B_k|, f_k)$
- Frequency at the frequency order *k*: f_k (Hz): $f_1, f_2, f_3, \ldots f_N$
- Volume of each element V_e (m^3)
- Maximum frequency order: N
- Multipliers/coefficients of the empiric formula: gamma, delta
- Number of elements: nelem

12-3-12-3. Equation A - 3

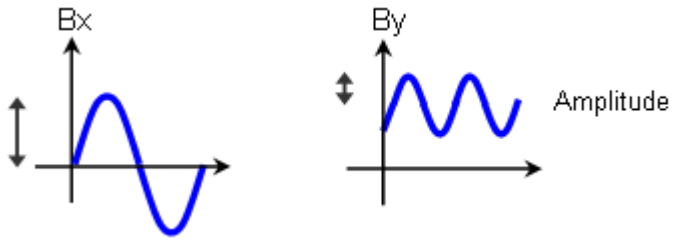
- Type of equation: Using experimental loss formula
- Type of magnetic flux density to be used: Apply loop
- Type of loss to be determined: Hysteresis loss

Apply the appropriate equation according to what item of magnetic flux density is to be referred when the result is evaluated.

■ **The equation to calculate loss of each magnetic flux density component:**

Apply the following equation when the amplitude of magnetic flux density is to be referred.

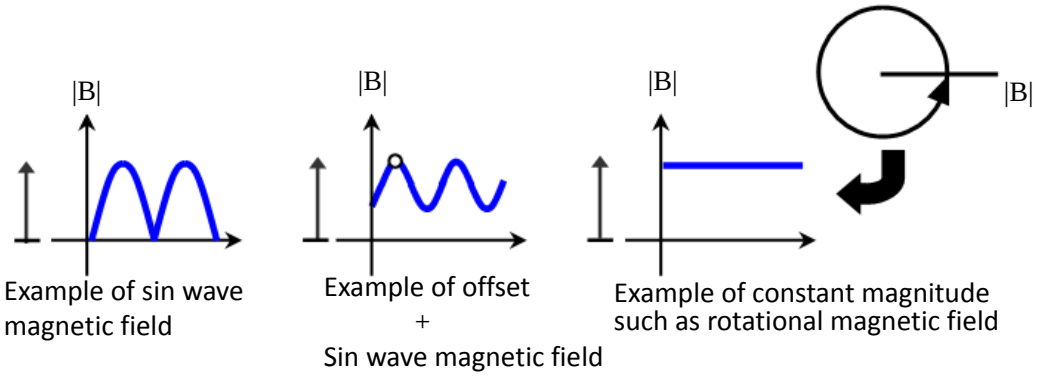
Equation A-3-1: $$Wh[W] = \sum_{e=1}^{nelem} \left\{ f \sum_{l=1}^{nloop} (Kh \times B_k^{\alpha}) \times V_e \right\}$$



■ Equation for calculating loss from the magnitude of magnetic flux density:

Apply the following equation when the maximum value of the magnetic flux density is to be referred.

Equation A-3-2: $$\mathrm{Wh[W]} = \sum_{e=1}^{nelem} \{ f \times Kh \times B_{max}^{\alpha} \times V_e \}$$



- B_k: Amplitude (k = 1 – *nloop*) of k-th loop for each component (X, Y, Z or r, theta, z) of magnetic flux density
- B_{max}: Maximum value of magnetic flux density (T)
- *nloop*: Number of loops
- Coefficients of hysteresis loss, *Kh*= constant, or $Kh(|B_k|, f_k)$
- Basic frequency f (Hz)
- Volume of each element V_e (m^3)
- Multipliers/coefficients of the empiric formula: alpha, beta
- Number of elements: nelem

12-3-12-4. Equation A - 4

- Type of equation: Using experimental loss formula.
- Type of magnetic flux density to be used: Maximum value
- Type of loss to be determined: Joule loss

The equation used to determine the loss is as follows.

Equation A-4: $$We[W] = \sum_{e=1}^{nelem} [\{Ke \times B_{max}^{\gamma} \times f^{\delta}\} \times V_e]$$

- Maximum value of magnetic flux density in the specified interval, B_{max} (T):
  - Determining the loss for each component of magnetic flux density:

    B_{max}= Maximum value of each component (X, Y, Z or r, theta, z) of the magnetic flux density at the interval.

  - Determining the loss from the amplitude of the magnetic flux density:

    $$B_{max} = \sqrt{B_1^2 + B_2^2 + B_{3max}^2}$$

    B_1, B_2 and B_3 are components (X, Y, Z or r, theta, z) of the magnetic flux density.

- Coefficients of hysteresis loss *Kh*, *Kh*= constant, or $Kh(|B_k|, f_k)$
- Coefficients of joule loss *Ke*: *Ke*= constant, or $Ke(|B_k|, f_k)$
- Basic frequency f (Hz)
- Volume of each element V_e (m^3)
- Multipliers/coefficients of the empiric formula: gamma, delta
- Number of elements: nelem

12-3-12-5. Equation B - 1

- Type of equation: Using loss data of material database
- Type of magnetic flux density to be used: FFT (frequency analysis)
- Type of loss to be determined: Hysteresis loss

  The equation used to determine the loss is as follows.

Equation B-1: $$Wh[W] = \sum_{e=1}^{nelem} \left\{ \sum_{k=1}^{N} \alpha(|B_k|) \times f_k \right\} \times V_e$$

- Magnetic flux density $|B_k|$ (T) at the frequency order k:
  - Determining the loss for each component of the magnetic flux density:

    $$|B_k| = \sqrt{B_k * B_k^*}$$

    B_k: Value (complex number) after FFT for each component (X, Y, Z, or r, q, z) of the magnetic flux density

    B_k^*: Complex conjugate numbers of B_k

  - Determining the loss from the amplitude of the magnetic flux density:

    $$|B_k| = \sqrt{B1_k * B1_k^* + B2_k * B2_k^* + B3_k * B3_k^*}$$

    $B1_k$, $B2_k$, $B3_k$: Value (complex number) after FFT for each component (X, Y, Z or r, theta, z) of the magnetic flux density

    $B1_k^*$, $B2_k^*$, $B3_k^*$: Complex conjugate numbers of $B1_k$, $B2_k$, $B3_k$

- Frequency at the frequency order k: f_k (Hz): $f_1, f_2, f_3, \ldots f_N$
- $a(|B_k|)$: Coefficient of magnetic flux density $|B_k|$ determined by the frequency separation method
- Volume of each element V_e (m^3)
- Maximum frequency order: N
- Number of elements: nelem

12-3-12-6. Equation B - 2

- Type of equation: Using loss data of material database
- Type of magnetic flux density to be used: FFT (frequency analysis)
- Type of loss to be determined: Joule loss

The equation used to determine the loss is as follows.

Equation B-2: $$We[W] = \sum_{e=1}^{nelem} \left\{ \sum_{k=1}^{N} b(|B_k|, f_k) \times f_k^2 \right\} \times V_e$$

- Magnetic flux density $|B_k|$ (T) at the frequency order k:
  - Determining the loss for each component of the magnetic flux density:

    $$|B_k| = \sqrt{B_k * B_k^*}$$

    B_k: Value (complex number) after FFT for each component (X, Y, Z, or r, theta, z) of the magnetic flux density.

    B_k^*: Complex conjugate numbers of B_k

  - Determining the loss from the amplitude of the magnetic flux density:

    $$|B_k| = \sqrt{B1_k * B1_k^* + B2_k * B2_k^* + B3_k * B3_k^*}$$

    $B1_k$, $B2_k$, $B3_k$: Value (complex number) after FFT for each component (X, Y, Z or r, theta, z) of the magnetic flux density.

    $B1_k^*$, $B2_k^*$, $B3_k^*$: Complex conjugate numbers of $B1_k$,$B2_k$, $B3_k$

- Frequency f_k (Hz) at the frequency order k: $f_1, f_2, f_3, \ldots f_N$
- $b(|B_k|, f_k)$: Magnetic flux density $|B_k|$ determined by the frequency separation method, coefficient at frequency f_k.
- Volume of each element V_e (m^3)
- Maximum frequency order: N
- Number of elements: nelem

12-3-12-7. Equation B - 3

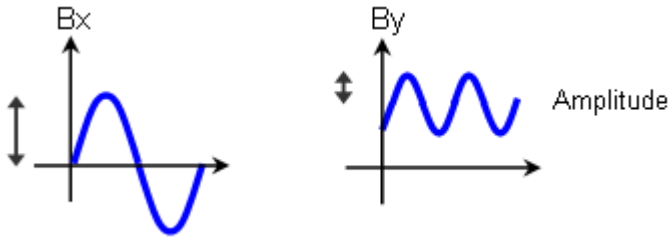
- Type of equation: Using the loss data of the material database.
- Type of magnetic flux density to be used: Apply loop
- Type of loss to be determined: Hysteresis loss

Apply the appropriate equation according to what item of magnetic flux density is to be referred when the result is evaluated.

■ The equation to calculate loss of each magnetic flux density component:

Apply the following equation when the amplitude of magnetic flux density is to be referred.

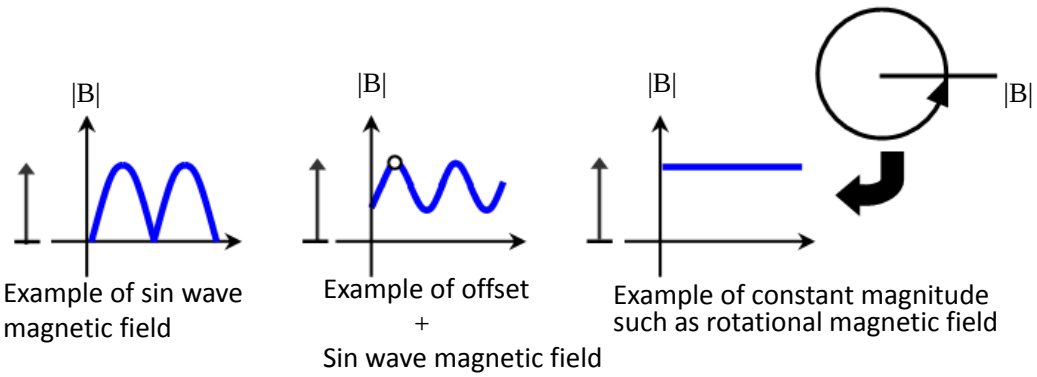
Equation B-3-1: $$Wh[W] = \sum_{e=1}^{nelem} \left\{ f \sum_{k=1}^{nloop} a(B_k) \times V_e \right\}$$



■ Equation for calculating loss from the magnitude of magnetic flux density:

Apply the following equation when the maximum value of the magnetic flux density is to be referred.

Equation B-3-2: $$Wh[W] = \sum_{e=1}^{nelem} \{f \times a(B_{max}) \times V_e\}$$



- B_k: Amplitude (k = 1 – *nloop*) of k-th loop for each component (X, Y, Z or r, theta, z) of magnetic flux density
- B_{max}: Maximum value of magnetic flux density (T)
- *a(B)*: Coefficient of magnetic flux density *B* determined by the frequency separation method
- *nloop*: Number of loops.
- Basic frequency *f* (Hz)
- Volume of each element V_e (m^3)
- Number of elements: nelem

12-3-12-8. Equation B - 4

- Type of equation: Using the loss data of the material database.
- Type of magnetic flux density to be used: Maximum value
- Type of loss to be determined: Joule loss

The equation used to determine the loss is as follows.

Equation B-4: $$We[W] = \sum_{e=1}^{nelem} [\{b(B_{max}, f) \times f^2\} \times V_e]$$

- Maximum value of magnetic flux density in the specified interval B_{max} (T):
  - Determining the loss for each component of the magnetic flux density:

    B_{max}= Maximum value of each component (X, Y, Z or r, theta, z) of the magnetic flux density at the interval

  - Determining the loss from the amplitude of the magnetic flux density:

    $$B_{max} = \sqrt{B_1^2 + B_2^2 + B_{3max}^2}$$

    B_1, B_2, B_3: Each component (X, Y, Z or r, theta, z) of the magnetic flux density.

- Basic frequency f (Hz)
- $b(B_{max}, f)$: Magnetic flux density B_{max} determined by the frequency separation method, coefficient at frequency f
- Volume of each element V_e (m^3)
- Number of elements: nelem

12-3-12-9. Equation C - 1

- Type of equation: A method in which the loss file is referred to, and specification is made separately for hysteresis loss and joule loss.
- Type of magnetic flux density to be used: FFT (frequency analysis)
- Type of loss to be determined: Hysteresis loss

The equation used to determine the loss is as follows.

Equation C-1: $$Wh[W] = \sum_{e=1}^{nelem} \left\{ \sum_{k=1}^{N} Wh_e(|B_k|, f_k) \times V_e \right\}$$

- Magnetic flux density $|B_k|$ (T) at the frequency order k:
  - Determining the loss for each component of the magnetic flux density:

    $$|B_k| = \sqrt{B_k * B_k^*}$$

    B_k: Value (complex number) after FFT for each component (X, Y, Z, or r, theta, z) of the magnetic flux density.

    B_k^*: Complex conjugate numbers of B_k

  - Determining the loss from the amplitude of the magnetic flux density:

    $$|B_k| = \sqrt{B1_k * B1_k^* + B2_k * B2_k^* + B3_k * B3_k^*}$$

    $B1_k$, $B2_k$, $B3_k$: Value (complex number) after FFT for each component (X, Y, Z or r, theta, z) of magnetic flux density.

    $B1_k^*$, $B2_k^*$, $B3_k^*$: Complex conjugate numbers of $B1_k$, $B2_k$, $B3_k$

- Frequency (Hz): at the frequency order k: $f_1, f_2, f_3, \ldots f_N$
- $Wh_e\left(B_k, \frac{1}{T}\right)$: Hysteresis loss density specified at the magnetic flux density B_k and the frequency f_k
- Volume of each element V_e (m^3)
- Maximum frequency order: N
- Number of elements: nelem

12-3-12-10. Equation C - 2

- Type of equation: A method in which the loss file is referred to, and specification is made separately for hysteresis loss and joule loss.
- Type of magnetic flux density to be used: FFT (frequency analysis)
- Type of loss to be determined: Joule loss

The equation used to determine the loss is as follows.

Equation C-2: $$We = \sum_{e=1}^{nelem} \left\{ \sum_{k=1}^{N} We_e(|B_k|, f_k) \times V_e \right\}$$

- Magnetic flux density $|B_k|$ (T) at the frequency order k:
  - Determining the loss for each component of the magnetic flux density:

    $$|B_k| = \sqrt{B_k * B_k^*}$$

    B_k: Value (complex number) after FFT for each component (X, Y, Z, or r, theta, z) of magnetic flux density.

    B_k^*: Complex conjugate numbers of B_k

  - Determining the loss from the amplitude of the magnetic flux density:

    $$|B_k| = \sqrt{B1_k * B1_k^* + B2_k * B2_k^* + B3_k * B3_k^*}$$

    $B1_k$, $B2_k$, $B3_k$: Value (complex number) after FFT for each component (X, Y, Z or r, theta, z) of magnetic flux density

    $B1_k^*$, $B2_k^*$, $B3_k^*$: Complex conjugate numbers of $B1_k$, $B2_k$, $B3_k$

- Frequency (Hz) at the frequency order, k: $f_1, f_2, f_3, \ldots, f_N$
- $We_e\left(B_k, \frac{1}{T}\right)$: Joule loss density specified at the magnetic flux density B_k and the frequency f_k
- Volume of each element V_e (m^3)
- Maximum frequency order: N
- Number of elements: nelem

12-3-12-11. Equation C - 3

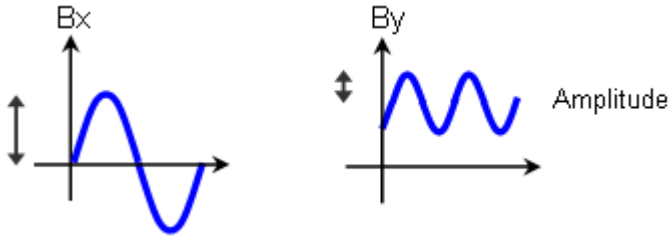
- Type of equation: A method in which the loss file is referred to, and specification is made separately for hysteresis loss and joule loss.
- Type of magnetic flux density to be used: Apply loop
- Type of loss to be determined: Hysteresis loss

Apply the appropriate equation according to what item of magnetic flux density is to be referred when the result is evaluated.

■ **The equation to calculate loss of each magnetic flux density component:**

Apply the following equation when the amplitude of magnetic flux density is to be referred.

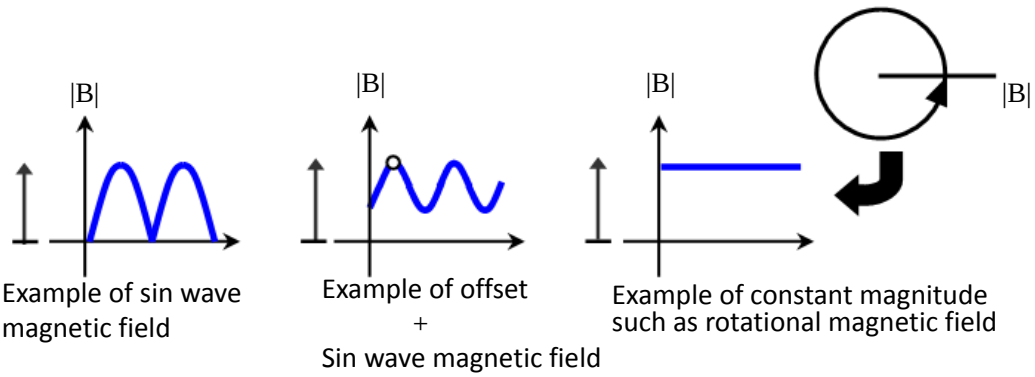
Equation C-3-1: $$Wh[W] = \sum_{e=1}^{nelem} \{Wh_e(B_{max}, f) \times V_e\}$$



■ Equation for calculating loss from the magnitude of magnetic flux density:

Apply the following equation when the maximum value of the magnetic flux density is to be referred.

Equation C-3-2: $$\mathrm{Wh[W]} = \sum_{e=1}^{nelem} \{\mathrm{Wh_e}(\mathrm{B_{max}}, \mathrm{f}) \times \mathrm{V_e}\}$$



- B_k: Amplitude (k = 1 – *nloop*) of k-th loop for each component (X, Y, Z or r, theta, z) of magnetic flux density
- B_{max}: Maximum value of magnetic flux density (T)
- $Wh_e(B, f)$: Hysteresis loss density specified at the magnetic flux density B and the frequency f
- *nloop*: Number of loops
- Basic frequency f (Hz)
- Volume of each element V_e (m^3)
- Number of elements: nelem

12-3-12-12. Equation C - 4

- Type of equation: A method in which the loss file is referred to, and specification is made separately for hysteresis loss and joule loss.
- Type of magnetic flux density to be used: Maximum value
- Type of loss to be determined: Joule loss

The equation used to determine the loss is as follows.

Equation C-4: $$We[W] = \sum_{e=1}^{nelem} [\{We_e(B_{max}, f)\} \times V_e]$$

- Maximum value of magnetic flux density in the specified interval B_{max} (T):
  - Determining the loss for each component of the magnetic flux density:
    B_{max}= Maximum value of each component (X, Y, Z or r, theta, z) of the magnetic flux density at the interval.
  - Determining the loss from the amplitude of the magnetic flux density:
    $$B_{max} = \sqrt{B_1^2 + B_2^2 + B_{3max}^2}$$
    B_1, B_2, B_3: Each component (X, Y, Z or r, theta, z) of the magnetic flux density.
- Basic frequency f (Hz)
- $We_e(B_{max}, f)$: Joule loss density specified at the magnetic flux density B_{max}, and the frequency f
- Volume of each element V_e (m^3)
- Number of elements: nelem

12-3-12-13. Frequency Separation Method

When you select the "using loss data of the material database" in the equations, JMAG-Studio calculates hysteresis loss and joule loss separately using a frequency separation method. For a frequency not included in the loss density data of the material database, JMAG-Studio uses a frequency separation method to complement the data.

The loss density data given by the material database include both hysteresis loss and joule loss. If we assume that hysteresis loss is proportional to the frequency and Joule loss to the square of it, the loss W at a magnetic flux density B is considered to show the following dependence on frequency.

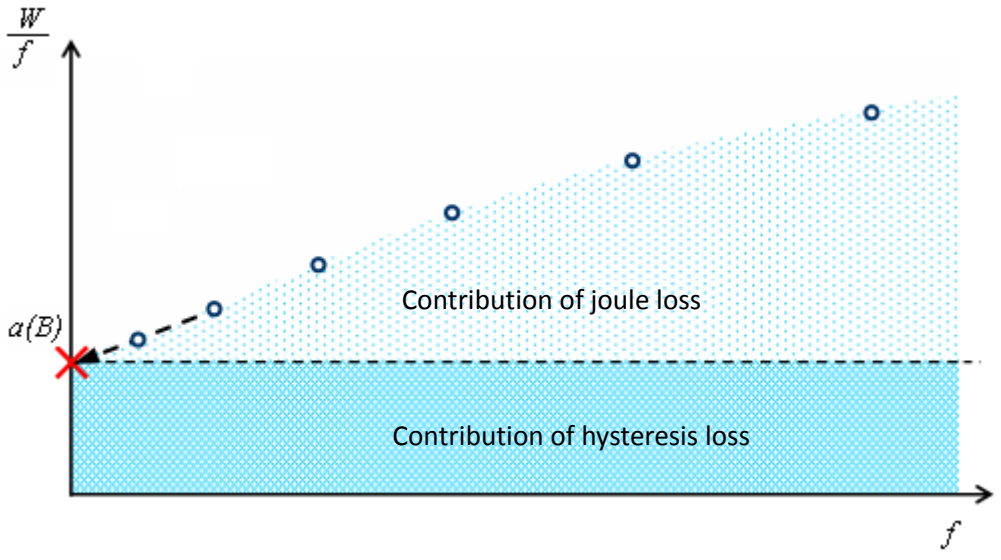
$$W = a(B) \times f + b(B, f) \times f^2$$

Dividing both sides of this equation by frequency f, we obtain

$$\frac{W}{f} = a(B) + b(B, f) \times f$$

Then after plotting data on the coordinate system with f as the transverse axis and $\frac{W}{f}$ as the longitudinal axis, an almost linear relation is obtained.

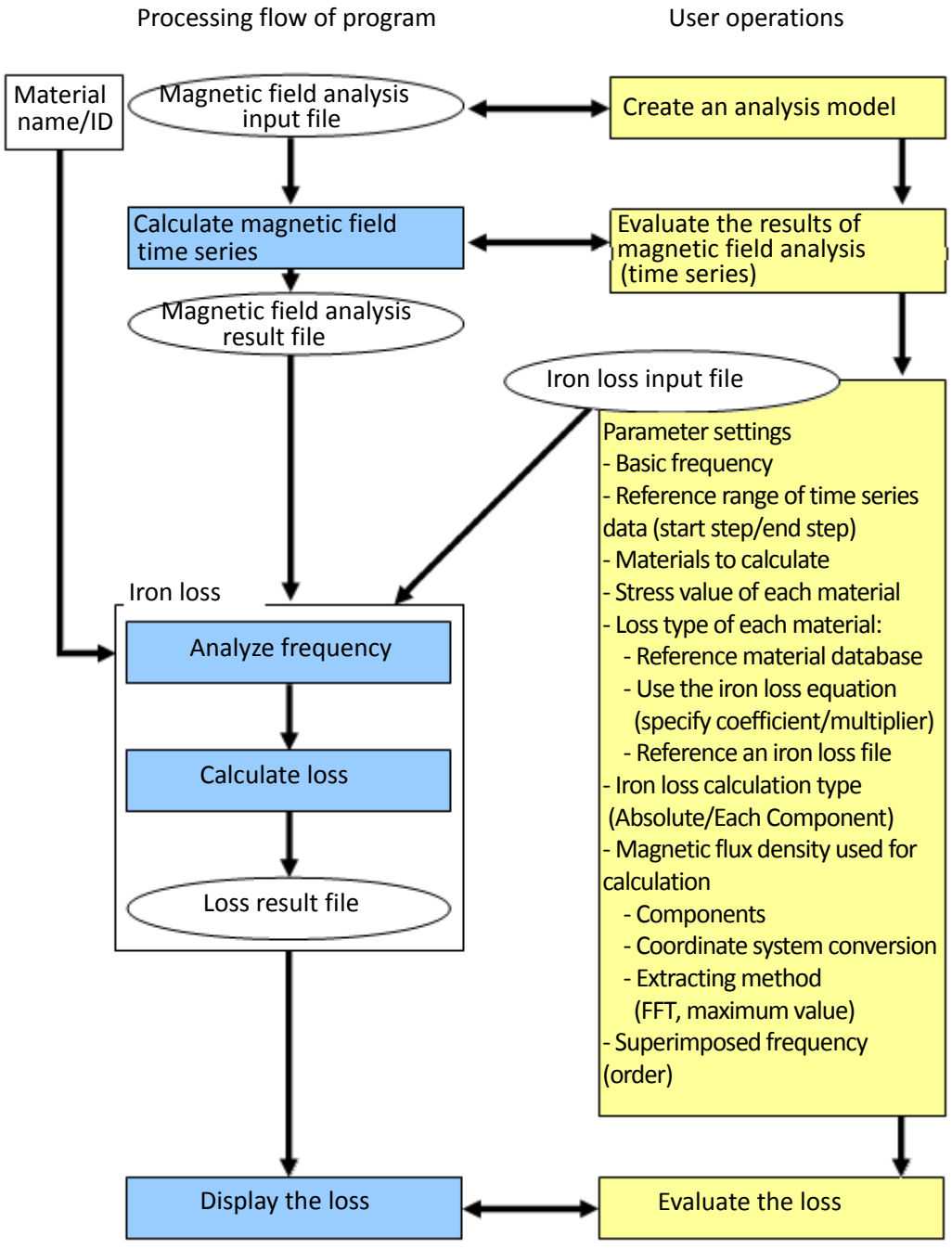
The intercept of the line is $a(B)$ and the slope is $b(B, f)$. However, since the relation slightly deviates from the line, $b(B, f)$ is considered to be frequency dependent.



Separation of hysteresis loss and joule loss

12-3-13. Procedure for Evaluating the Loss

The loss calculation in JMAG is processed by following the procedure below.


Processing flow of program
User operations
Material name/ID
Magnetic field analysis input file
Create an analysis model
Calculate magnetic field time series
Evaluate the results of magnetic field analysis (time series)
Magnetic field analysis result file
Iron loss input file
Parameter settings
- Basic frequency
- Reference range of time series data (start step/end step)
- Materials to calculate
- Stress value of each material
- Loss type of each material:
- Reference material database
- Use the iron loss equation (specify coefficient/multiplier)
- Reference an iron loss file
- Iron loss calculation type (Absolute/Each Component)
- Magnetic flux density used for calculation
- Components
- Coordinate system conversion
- Extracting method (FFT, maximum value)
- Superimposed frequency (order)
Iron loss
Analyze frequency
Calculate loss
Loss result file
Display the loss
Evaluate the loss

Procedures for calculating loss

12-3-13-1. Performing Frequency Analysis to Evaluate Loss

1. **Transient response magnetic field analysis will be executed.**
2. **The intervals of time-series data of the magnetic flux density used for the loss calculation are determined from the Basic frequency, reference start step, and reference end step.**
3. **Time-series data of magnetic flux density within the interval are extracted for each material of the transformation object in consideration of the coordinate system transformation and rotational motion.**
4. **To perform FFT analysis, 2^N data points are required. Spline interpolation scheme is used to create 2^N more equally spaced data points than the specified number of data points.**

   For example, if 24 data points are specified, they are interpolated to increase to 32 data points.
5. **FFT analysis is performed. It creates $2^N/2$-1 frequency components other than direct current components. The lowest frequency is the Basic frequency specified.**
6. **The loss is calculated for each element by substituting the above obtained magnetic flux density into the equation.**
7. **The loss for each specified order of frequency and the sum of losses over all frequencies are output to the plot or Jplot file.**

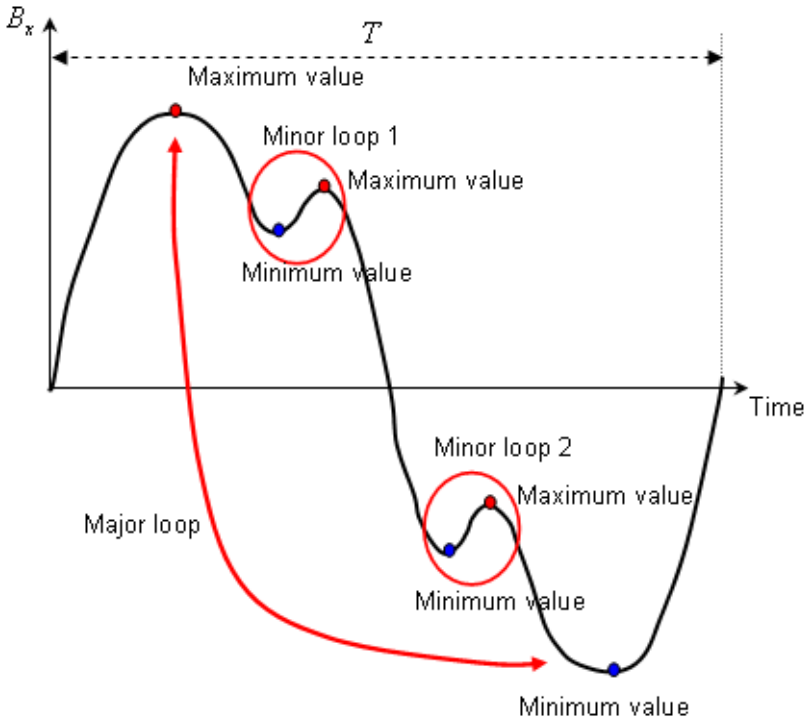
12-3-13-2. Procedure to Evaluate Loss from the Maximum Value of Magnetic Flux Density (Hysteresis Loss)

To evaluate the loss by using the maximum value of magnetic flux density, Joule loss is calculated from the maximum value at the specified interval of magnetic flux density. However, in calculating the hysteresis loss, the minor loop should be considered as well as the maximum value.

1. **Transient response magnetic field analysis will be executed.**
2. **The intervals of time-series data of magnetic flux density used for the loss calculation are determined from the basic frequency, reference start step, and reference end step.**
3. **Time-series data of magnetic flux density within the interval are extracted for each material of the transformation object in consideration of the coordinate system transformation and rotational motion.**
4. **The maximum value and minimum value of magnetic flux density are searched at the interval, and whether they belong to either the major loop or the minor loop is determined. In the example shown below, there are two minor loops besides the one major loop.**
   The amplitude of a loop is given by |Maximum value - Minimum value | / 2.
5. **The loss due to a minor loop is determined only by the amplitude. The difference in the loss along with the position of a minor loop in the main loop is not considered.**
6. **If data is not periodic within the interval, the hysteresis loop cannot be counted. Specify basic frequency, reference start step, and reference end step for magnetic flux density to be periodic within the interval. For 1/2- or 1/4-cycle data, use the expanded function of periodicity.**

   To use the expanded function of periodicity, set the [Option] tab in the [Calculate Iron Loss] dialog box.
7. **The loss is calculated for each element by substituting the above obtained magnetic flux density in the equation.**

8. The sum of losses over all loops is output to the plot or Jplot file.



Hysteresis loop counting

12-3-13-3. Procedure to Evaluate Loss from the Maximum Value of Magnetic Flux Density (Joule Loss)

1. **Transient response magnetic field analysis will be executed.**
2. **The intervals of time-series data of magnetic flux density used for the loss calculation are determined from the basic frequency, reference start step, and reference end step.**
3. **Maximum magnetic flux density of time-series data within the interval of magnetic flux density is determined for each material of the transformation object.**
4. **The loss is calculated for each element by substituting the above-obtained frequency and magnetic flux density in the equation.**
5. **Joule loss at a specified basic frequency is output to the plot or Jplot file.**

12-3-14. User Subroutine *iron_str*

12-3-14-1. Settings

Defines the stress dependency of the iron loss properties. The functions required to access the iron loss properties data is provided in addition to *iron_str*. In a user subroutine, you can create the flux density and iron loss properties data at each frequency with the input of principal stress for each element.To use *iron_str*, select the [Use User Subroutine] check box and then enter the ID of a negative integer for [User Subroutine ID] as you specify the iron loss property from the material database. *iron_str* can only be used for the material properties defined as UserSteel that have iron loss properties with stress dependency.

12-3-14-2. Arguments

Variable	Details	I/O
u_mid	User Subroutine ID	I
CIronsLossFile* i_Plf	Pointer of iron loss properties	I
e_id	Element ID	I
m_id	Material ID	I
Pstrs[3]	Principal stress (I), principal stress (II), principal stress (III) (Pa)	I
SStrs[3]	Shear stress, XY component, YZ component, ZX component (Pa)	I
Strs3[3]	Stress, X component, Y component, Z component (Pa)	I

12-3-14-3. Function

```
int GetNumS(CIronsLossFile* i_Plf);
```
Gives the number of stress values contained in the iron loss properties data.

```
double GetS(CIronsLossFile* i_Plf,int i_strs);
```
Gives the stress value in the iron loss properties data when entering the order ID i_strs.

```
double GetF(CIronsLossFile* i_Plf,int i_freq);
```
Specifies the frequency in the iron loss properties data when entering the order ID i_freq.

```
int GetNumBW(CIronsLossFile* i_Plf,int i_strs, int i_freq);
```
Specifies the number of points in the point sequence of flux density and loss in the iron loss properties data when entering i_strs, the order ID of stress and i_freq, the order ID of frequency.

```
double GetB(CIronsLossFile* i_Plf,int i_strs, int i_freq, int
i_BW);
```
Specifies the flux density in the iron loss properties data when entering i_strs, the order ID of stress, i_freq, the order ID of frequency, and i_BW the order ID of the point sequence of flux density versus loss.

```
double GetW(CIronsLossFile* i_Plf,int i_strs, int i_freq, int
i_BW);
```
Specifies the loss value in the iron loss properties data when entering i_strs, the order ID of stress, i_freq, the order ID of frequency, and i_BW the order ID of the point sequence of flux density versus loss.

```
bool SetFrequency(CIronsLossFile* i_Plf, int i_freq, double
Fval);
```
Sets Fval, the frequency value for i_freq, the order ID of the frequency. Returns false, if failed.

```
bool SetFrequencyTable(CIronsLossFile* i_Plf, int i_freq, int
num);
```
Reserves a memory space for the point sequence of flux density versus loss at a frequency when entering i_freq, the order ID of frequency and num, the number of points in the point sequence of flux density versus loss. Returns false, if failed.

```
bool SetB(CIronsLossFile* i_Plf, int i_freq, int i_BW, double
Bval);
```

Specifies Bval, the flux density value when entering i_freq, the order ID of the frequency and i_Bw, the order ID of the point sequence of flux density versus loss. Returns false, if failed.

```
bool SetW(CIronsLossFile* i_Plf, int i_freq, int i_BW, double
Wval);
```

Specifies Wval, the loss value when entering i_freq, the order ID of the frequency and i_BW, the order ID of the point sequence of flux density versus loss. Returns false, if failed.

```
bool MakeIronStrTable(CIronsLossFile* i_Plf, int numf);
```

Reserves the memory space for the iron loss properties data when entering numf, the number of frequencies. Returns false, if failed.

12-3-14-4. Example

Define the stress dependency on the second component of the principal stress. From the iron loss properties data, obtain the two properties adjacent to the principal stress (frequency - flux density - loss), and linearly interpolate between the two principal stresses to defined the new iron loss property (frequency - flux density - loss). The iron loss properties with stress dependency must have the same number of frequencies and points in a point sequence of flux density versus loss at every stress value.

```
#include <stdio.h>
#include <math.h>
#include "usrsub_ironloss.h"

void iron_str(void* i_Plf,
                              int e_id,
                              int u_mid,
                              const double PStrs[3],
                              const double SStrs[3],
                              const double Strs3[3])
{
    double dStr = 0.0;
    int ip1,ip2,idx1,idx2;
    double S0,S1,S2,ss1,ss2;
    int mode = 0;
    int i,j,k;
    int numf;
    int num0,num1,num2,numt;
    double b0,b1,b2;
    double w0,w1,w2,w01,w02;
    double fval;
    double btmp = 0.0;
    int nums;
    double i_stress = PStrs[2];

    nums = GetNumS(i_Plf);

    S0 = GetS(i_Plf,0);
    dStr = i_stress - S0;
    if(dStr<0.0){
        ip1 = 0;
        S1 = GetS(i_Plf,ip1);
        mode = 1;
    }
    S0 = GetS(i_Plf,nums-1);
    dStr = i_stress - S0;
    if(dStr>0.0){
        ip1 = nums-1;
        S1 = GetS(i_Plf,ip1);
        mode = 1;
    }
```

```
for(i=1; i<nums; i++){
    dStr = i_stress - GetS(i_Plf,i);
    if(dStr<0.0){
        ip1 = i-1;
        ip2 = i;
        mode = 2;
        S1 = GetS(i_Plf,ip1);
        S2 = GetS(i_Plf,ip2);
        break;
    }
}
numf = GetNumF(i_Plf);
MakeIronStrTable(i_Plf,numf);

if(mode == 1){
    for(i=0;i<numf;i++){
        fval = GetF(i_Plf,i);
        SetFrequency(i_Plf,i,fval);
        num0 = GetNumBW(i_Plf,ip1,i);
        SetFrequencyTable(i_Plf,i,num0);
        for(j=0;j<num0;j++){
            b0 = GetB(i_Plf,ip1,i,j);
            w0 = GetW(i_Plf,ip1,i,j);
            SetB(i_Plf,i,j,b0);
            SetW(i_Plf,i,j,w0);
        }
    }
}else{
    for(i=0;i<numf;i++){
        fval = GetF(i_Plf,i);
        SetFrequency(i_Plf,i,fval);
        num1 = GetNumBW(i_Plf,ip1,i);
        num2 = GetNumBW(i_Plf,ip2,i);
        b1 = GetB(i_Plf,ip1,i,num1-1);
        b2 = GetB(i_Plf,ip2,i,num2-1);
        if(b1<=b2){
            num0 = num1;
            numt = num2;
            idx1 = ip1;
            idx2 = ip2;
            ss1= S1;
            ss2= S2;
        }else{
            num0 = num2;
            numt = num1;
            idx1 = ip2;
            idx2 = ip1;
            ss1= S2;
            ss2= S1;
        }
        SetFrequencyTable(i_Plf,i,num0);
        SetB(i_Plf,i,0,0.0);
        SetW(i_Plf,i,0,0.0);
```

```
                    for(j=1;j<num0;j++){
                        b0 = GetB(i_Plf,idx1,i,j);
                        w01 = GetW(i_Plf,idx1,i,j);
                        for(k=1;k<numt;k++){
                            b2 = GetB(i_Plf,idx2,i,k);
                            btmp = b2 - b0;
                            if(fabs(btmp) < 1e-30){
                                break;
                            }
                            if(k==(numt-1)){
                                break;
                            }
                        }
                        b1 = GetB(i_Plf,idx2,i,k-1);
                        w2 = GetW(i_Plf,idx2,i,k);
                        w1 = GetW(i_Plf,idx2,i,k-1);
                        w02 = w1 + (w2-w1)*(b0-b1)/(b2-b1);
                        w0 = w01 + (w02-w01)*(i_stress-S1)/(S2-S1);
                        SetB(i_Plf,i,j,b0);
                        SetW(i_Plf,i,j,w0);
                    }
                }
            }
        }
```

12-4. Calculate Particle Trajectory (JMAG-Studio)

12-4-1. Function

This tool calculates the paths of charged particles flying in magnetic fields and electric fields as described by the data in the PLOT file obtained by magnetic field and electric field analyses using JMAG-Studio. The paths obtained from particle trajectory are superimposed on the referenced PLOT file and displayed in the model.

Magnetic field and electric field forces exerting on charged particles can be calculated from the velocity of charged particles, and magnetic field density and electric fields acting on charged particles. However magnetic flux density and electric fields calculated by magnetic field and electric field analyses using JMAG-Studio are those evaluated in an element unit.

Then, using magnetic flux density and the electric field for each element calculated by the integral method, this program evaluates and analyzes forces exerting on charged particles.

12-4-2. Important Notes Regarding the Use of This Function

- Charged particles cannot fly within magnets, current density elements, or inductors.
- Determinations of inside or outside the magnet, current density elements, or inductors are not performed because of the calculation time.
- Calculation results where a particle plunges into magnets, current density elements, or inductors are not reliable.
- Analysis PLOT files are compatible with 3D magnetic field analysis (ST and TR) and electric field analysis (EL).

12-4-3. Procedures for Executing the Tool

1. **From the menu bar in JMAG-Studio, select [Tools] > [Calculate Particle Trajectory].**

   The Parameter for [Calculate Particle Trajectory] dialog box will appear.

2. **Specify the following items.**

   - [Magnetic Field Analysis Result File]: Specify the PLOT file name of the magnetic flux density or the electric field density that is used for orbit analysis

     If multiple file names are specified, calculations considering the contribution from multiple models can be performed. However the mutual relation between multiple models cannot be considered.

     When specifying a file name, specify the step number in a file to be applied for orbit analysis. Assign "1" for static analysis.

     To use the PLOT file obtained by magnetic field analysis, the results of both the magnetizing vector and the current density had to have been output. Note that the precision of an orbit calculation depends on the size of the magnetizing vector element, the current density element, and the external current (beam) element.

     To use the PLOT file obtained by electric field analysis, the results of the internal charges had to have been output. The precision of an orbit calculation depends on the size of the dielectric elements and elements including the nodes where voltage is assigned.
   - [Output File]: Set the name of the PLOT file to be output in the particle trajectory.
   - [Symmetry Boundary Condition]: Specify symmetry of the model against the X-Y, Y-Z, and Z-X planes in the PLOT file of magnetic field analysis.

     Even with multiple files specified, only one symmetry can be set. You should therefore use the same symmetry for multiple files when considering multiple files.
   - [Periodic Boundary Condition]: Set the rotational periodicity of the model in the PLOT file of magnetic field analysis.

     Specify "Angle of 360/periodicity" for the [Number of Copies].

     For example: 90-degree periodicity --> the [Number of Copies] is 4.

     Even with multiple files specified, only one symmetry can be set. You should therefore use the same symmetry for multiple files when considering multiple files.

     Translational periodicity cannot be taken into consideration.

- [Number of Integration Points]: Specify the integral order of numeric integration.

  In the current version, the value for a tetrahedron is automatically determined from that of a hexahedron as a dummy.

  The recommended value at sub-division is first order (single point integration).

  The higher the integral precision specified, the higher the precision that can be achieved, however, the calculation requires more time.

  It is practical that integral precision is kept low, and around nearby elements, higher precision is maintained by sub-division at intersection.
- [Subdivision at Intersection]: In order to maintain integration accuracy in the vicinity of the magnetizing vector element and electric current density element, sub-divide the elements hypothetically.
  - [Maximum Level of Division]: When the [Maximum Level of Division] is 0, sub-division is not performed.
  - [Base Distance]: When the "distance between the evaluation position and the integration point" is closer than the "Element Length x [Base Distance]", the sub-division of elements is performed.
- [Initial Position]: Specify the initial position of the particle at the end of the orbit calculation. The initial position is specified with the coordinate range and number of divisions.

  To fix the component value, specify "0" for the [Number of Division].

  When the number of divisions is "0", only the minimum value of the component is considered.
- [Initial Velocity]: Specify the initial position of the particle at the start of the orbit calculation.

  The direction vector of velocity is specified with the coordinate range and number of divisions in the same way as the initial position.

  When a negative value is specified as the magnitude of velocity, the direction vector of velocity is treated as a velocity vector, and the magnitude is not considered.
- [Mass (kg)]: Specify the mass of charged particles.
- [Charge (C)]: Specify the electric charge of charged particles.
- [Delta t (sec)]: Specify the width of time steps used for solving equations of motion.
- [Output Step Interval]: When the number of steps is high, output can be made with some steps being skipped.

- [Condition to Finish Analysis]: Specify conditions to terminate the orbit calculation. When specifying the condition by step, select the [Specify by Step] radio button, and enter the number of analysis steps.
  When specifying the condition by the flying range of particles, select the [Specify by Range] radio button, and specify the movable range. The units of movable range are (m).
- [Output Trajectory File]: Specify how to output the trajectory data file. If the check box is selected, all position data is output with the file name of "particle_orbit.txt" in addition to the results file.

3. Click [Run].

12-4-4. Output Items

- Particle velocity (m/s)
- Nodal force (N)
- Magnetic flux density (T)
- Path of article (output in beam elements)
- Path of article (output in a separate text file)

12-5. Calculate Field Value (JMAG-Studio)

12-5-1. Function

The Calculate Field Value tool (CALCB) calculates magnetic flux density and electric fields at a specified point in the air based on the PLOT file data of magnetic field analysis or electric field analysis.

Since each physical quantity is calculated per element in FEM analyses, the mesh needs to be made finer in order to accurately evaluate flux density and electric field in the air region, requiring more time for the calculation. CALCB allows you to draw smooth lines in graphs by using an integral method even if the mesh in the air region is coarse in the reference PLOT file.

In a normal 2D FEM analysis with BEM, flux density or electric filed in the air region which is not meshed cannot be plotted in a graph. However, using CALCB makes it possible to draw the graph of such data.

12-5-2. Important Notes Regarding the Use of This Function

12-5-2-1. Analysis Models

- 2D analysis with triangular or quadrilateral elements, and 3D analysis with tetrahedral, pentahedron (prism), or hexahedron elements are supported.
- This tool cannot be used for axisymmetric models.
- X-Y, Y-Z, and Z-X planes can be dealt with as symmetric or natural boundaries.
- Rotational periodic boundary (only one) can be taken into account in calculation.
- Models with translational periodicity or 3D analyses of equivalent 2D problems cannot be calculated accurately with this tool.

12-5-2-2. Files Required for Analysis

One of the following files is needed to run an analysis.

■ PLOT file for the magnetic field analysis

- Magnetization and current density need to be included in the file. Make sure the setting of [Control of Output Item] in the analysis control condition is correct. The magnetomotive force such as current or magnetization that is not contained in the PLOT file will not be taken into account in the integral method electromagnetic analysis.

- The flux density set for the external field condition in the magnetic field analysis cannot be included in the integral method electromagnetic analysis.
- The accuracy of integral method electromagnetic analysis depends on magnetization, current density, and current (beam) contained in the PLOT file for the magnetic field analysis. Adjust the mesh density so that the magnetization, current density, and current (beam) at the target point will be calculated with high accuracy.
- Make the mesh in the magnetic material as fine as possible if you want to calculate the flux density around the material accurately in the integral method electromagnetic analysis.

■ PLOT file for the electric field analysis

- An internal charge (either node charge, line charge density, face charge density, or polarization vector) needs to be output. The [Control Output Items] in the analysis control condition needs to be set carefully for the electric field analysis. The integral method calculator tool cannot be used to calculate an electric field using surface charge.
- The accuracy of integral method electromagnetic analysis depends on internal charge contained in the PLOT file for the electric field analysis. Adjust the mesh density for the region specified as dielectric or electric potential so that the internal charge at the target point will be calculated with high accuracy.

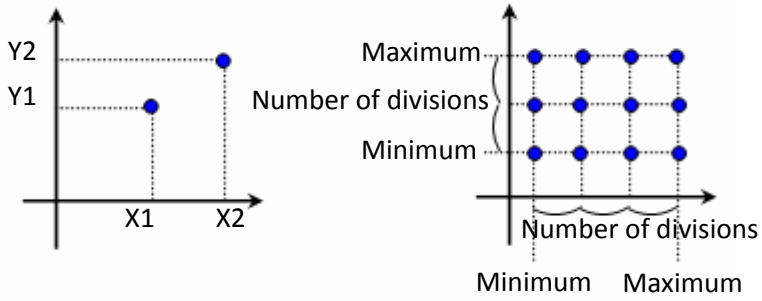
12-5-2-3. Notes on Analysis

- Set the analysis region in the air.
- For numerical integration precision, you can specify up to 10th order for quadrilateral element and hexahedron element (pentahedron element), and the integral order for triangle and tetrahedron varies with the specification of quadrilateral and hexahedron, but the maximum is 5th order.
- For subdividing elements, the following parameter values are recommended.
  - The number of integral points: 1
  - Subdivision base: 2.0
  - Maximum level: A value can be entered. For example, a value of about 5 to 7 is sufficient.
  - Physical quantity when subdividing: No interpolation is made, but element values are used.
- When setting the calculation points, select the coordinate system first. The following three types of coordinate system can be selected.
  - Local coordinate system (orthogonal [x (m), y (m), z (m)] *default

- Local coordinate system (cylindrical [r (m), θ (deg), z (m)])
- Local coordinate system (sphere [r (m), θ (deg), ϕ (deg)])

Next, select the setting method for the coordinates of the calculation points, and set the parameters while paying attention to the units. The following two types of setting method for coordinates are available.

- Method that directly sets the coordinates of the calculation points as a sequence of points (first axis, second axis, third axis)
- Method that fixes the calculation points based on the set regions and their number of divisions.



Point sequence Specify region

12-5-3. Procedures for Executing the Tool

1. From the menu bar in JMAG-Studio, select [Tools] > [Calculate Field Value].

The [Calculate Field Value] dialog box will appear.

2. Specify the following items.

- [Magnetic Field Analysis Result File]: Set the analyzing PLOT file of the magnetic field analysis or electric field analysis.

  If multiple files are specified, calculations considering the contribution from multiple models can be performed. But Mutual relation between multiple models cannot be considered.

  When specifying a file, specify the step number in a file to be applied for orbit analysis.Assign 1 for static analysis.

  Select or clear the following check boxes as required.

  - [Find by Suffix]: When this check box is selected, multiple result files can be specified. When files named “abc1.lot” or “abc2.plot” are specified, specify “abc” as the suffix.
  - [Show Full Path]: When this check box is selected, the full path of the referenced PLOT file is displayed in the list box.

  - [Use Multiple Results]: When multiple PLOT files or multiple steps are specified, specify whether or not to add magnetic flux density and electric field to the items output.
- [Output File]: Set the name of the PLOT file to be output in the integral method calculator.
- [Symmetry Boundary Condition]: Set the symmetry of the model with respect to the X-Y, Y-Z, and Z-X surfaces in the PLOT file of the magnetic field and electric field analyses. Even if multiple files are specified, the number of symmetries that can be set is only one. Thus, when considering multiple files, those files having the same symmetry should be considered for setting the symmetry.
- [Periodic Boundary Condition]: Set the rotational periodicity of the model in the PLOT file of the magnetic field analysis. Even if multiple files are specified, the number of periodicities that can be set is only one. Thus, when considering multiple files, those files having the same symmetry should be considered for setting periodicity.

  Translational periodicity cannot be considered. This setting is not applicable to the PLOT file of the electric field analysis.
  - [Number of Divisions]: Specify "360/Periodic angle".

    Example: Period of 90 degrees – [Number of Divisions]: 4.
- [Number of Integration Points]: Set the integral order of numerical integration. In the current version, the value for a tetrahedron is automatically substituted with that for a hexahedron. The recommended value for sub-dividing is first order (single point integration). The higher the integral accuracy is specified, the higher the accuracy that can be achieved; however, more calculation time is required. The practical way is to lower the integral accuracy and thereby keep the accuracy higher around elements in the sub-dividing.
- [Subdivision at Intersection]: In the vicinity of magnetizing vector element and electric current density element, in order to maintain integration accuracy, the elements are sub-divided hypothetically.
  - [Maximum Level of Division]: When this item is 0, sub-division is not conducted.
  - [Base Distance]: When the "distance between the evaluation position and the integration point" is closer than the "Element Length x [Base Distance]", the sub-division of elements is performed.
- [Type]: Specify the type of the coordinate system to be used.
  - Orthogonal coordinate system
  - Cylindrical coordinate system
  - Polar coordinate system

- [Local Coordinate System]: Specify the type of the local coordinate system to be used. Click [Setting] of the [Local Coordinate System] group box, and specify the next items.
  - [Center] of [Local Coordinate System]: Specify the origin of the local coordinates in specifying the local coordinate system.
  - [A Point on X-axis] of [Local Coordinate System]: Specify the X-direction of the orthogonal coordinates in specifying the local coordinate system.
  - [A Point on the XY-plane] of [Local Coordinate System]: Specify a point on the X-Y plane of the orthogonal coordinates in specifying the local coordinate system.
  - [A point on Axis] of [Rotation of Local Coordinate System]: Specify a point on the rotation axis when rotating the local coordinate system in specifying the local coordinate system.
  - [A Point on Vector] of [Rotation of Local Coordinate System]: Specify a rotation axis vector when rotating the local coordinate system in specifying the local coordinate system.
  - [Angle (deg)] of [Rotation Angle]: Specify a rotation angle when rotating the local coordinate system in specifying the local coordinate system.
- [Calculation Point]: Specify the method for specifying calculation point.
  - When Method for specifying calculation point is [Point sequence]: Specifying calculation point-by-point sequence.

    Click [Setting] of the [Point Sequence] group box and open the [Point Sequence List] dialog box. Each item of the point sequence to be set here is as follows

    ```
    1; First axis coordinate
    2; Second axis coordinate
    3; Third axis coordinate
    ```

    Note that the units differ as follows, based on the choice made in [Type].
    - Orthogonal coordinate system: first axis: (m), second axis: (m), third axis: (m)
    - Cylindrical coordinate system: first axis: (m), second axis: (deg), third axis: (m)
    - Polar coordinate system: first axis: (m), second axis: (deg), third axis: (deg)
  - When Method for specifying calculation point is [Region]: Set each parameter
    - Range of axis 1(Specifying calculation point by region): Range in the 1st axis direction of the local coordinate system. The unit is (m).

- Range of axis 2: Range in the 2nd axis direction of the local coordinate system.

  When the [Orthogonal Coordinate System] is selected in [Type], the unit is (m).

  When [Cylindrical Coordinate System] or [Polar Coordinate System] is selected, the unit is (deg).
- Range of axis 3: Range in the 3rd axis direction of the local coordinate system.

  When the [Orthogonal Coordinate System] or the [Cylindrical Coordinate System] is selected in [Type], the unit is (m). When [Polar Coordinate System] is selected, the unit is (deg).

3. Click [Calculate].

■ **Exporting and loading parameter files:**

- **To export parameter files, click [Export Parameter File], open the [Save As] dialog box, and after specifying the name, click [Save].**

  Electromagnetic field analysis parameter file (*.clb) of an arbitrary point will be exported.

- **To Import parameter files, click [Import Parameter File], open the [Open] dialog box, and specify a point electromagnetic field analysis parameter file (*.clb) from the list.**

12-5-4. Output Items

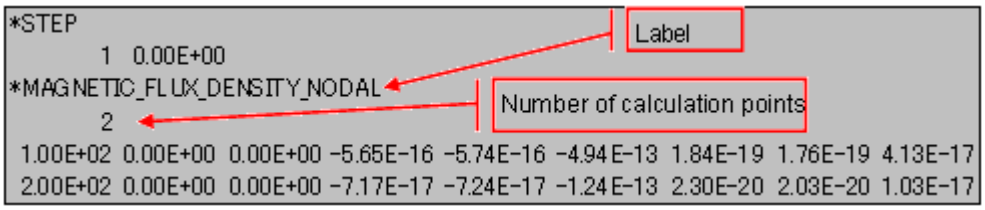
- Magnetic density (T)
- Electric field (V/m)

12-5-5. Output Files

- PLOT file
- Comma separated text file (*.csv)

  Output with comma-delimited.

  sample:



  The calculated values differ as follows based on the type of coordinate system (Lines 5 and 6 in the sample above).

  - (For the orthogonal coordinate system, the local coordinate system is also included)

    id (serial number), x, y, z, Bx, By, Bz, Ex, Ey, Ez:
  - (For the cylindrical coordinate system, the local coordinate system is also included)

    id (serial number), r, theta, z, Br, B-theta, Bz, Er, E-theta, Ez:

12-5-6. Analysis Control File Format

See the following description and control file samples contained in the check file that accompany the module.

In the analysis control file, lines beginning with " ' " are comments.

1st line	Version number
2nd line	Symmetry boundary conditions (int symYZ, int symZX, int symXY) Setting of symmetry boundary conditions sym : 1 Symmetry boundary 0 None -1 Natural boundary
3rd line	Flag for using rotational periodic boundary conditions (int periodic FLAG) Use or nonuse of periodic boundary FLAG : 0 Nonuse 1 Periodic boundary -1 Anti-periodic boundary

(If Periodic boundary or Anti-periodic boundary is used, the following exist. With Non-use, the following do not exist.)

[3-1 line]	A point through which the axis of rotational periodic boundary passes
[3-2 line]	Axial vector of rotational periodic boundary (double vx, double vy, double vz)
[3-3 line]	Number of division (360 degrees/rotation angle) (int numDIV)
4th line	Add results together flag FLAG: 0 Totaling 1 Not totaling
5th line	Number of PLOT file (int numFile)

(6th and 7th lines are repeated to the number of PLOT file.)

6th line	Name of PLOT file (char filename [ ]) (Enclosed in double quotation marks "".)
7th line	Objective step number (int stepNo) Specifies the step number of the PLOT file to be calculated.
8th line	Number of integral points (int numINT, int numINTt) 1-10, 1-6 The larger the number is, the higher the precision is, but calculation volume increases accordingly

9th line	Specifies sub-division (int MAX_LEVEL, double Sub_Divide_go) Nearby elements to be integrated: Virtual sub-division increases the integral precision. Maximum sub-division level, Sub-division reference.
10th line	Center of local coordinate system (double ox, double oy, double oz) Set the origin of the local coordinates in specifying the local coordinate system.
11th line	X' vector (double vx, double vy, double vz) Set the X-direction of the orthogonal coordinates in specifying the local coordinate system.
12th line	A point on X'-Y' plane (double x1, double y1, double z1) Set a point on the X'-Y' plane of orthogonal coordinate in specifying the local coordinate system.
13th line	A point on rotation axis in the local coordinate system (double rox, double roy, double roz) Set a point on the rotation axis when rotating the local coordinate system specified by lines 10 to 12.
14th line	Rotation axis vector in the local coordinate system (double rax, double ray, double raz) Set a rotation axis vector when rotating the local coordinate system specified by lines 10 to 12.
15th line	Rotation angle of the local coordinate system (double angle) Set a rotation axis vector when rotating the local coordinate system specified by lines 10 to 12.
16th line	Local Coordinate System Type (int lcodType) Type of local coordinate system used. 0: Orthogonal Coordinate System 1: Cylindrical Coordinate System 2: Polar Coordinate System
17th line	Method for specifying calculation point (int specifyType) Method for specifying calculation point. 0: By point sequence 1: By region

(Specifying calculation point by point sequence)

18th line	The number of calculation points (int numCalp) Specify the number of calculation points
	Coordinates of calculation point (int i, double p1, double p2, double p3)
19th line	Specify the serial number and coordinates of the calculation points (serial number, 1st-axis coordinate, 2nd-axis coordinate, 3rd-axis coordinate).

(Specifying calculation point by region)

	1st-axis range (double xmin, double xmax, int xdiv)
18th line	Range against the 1st-axis direction in the local coordinate system. (Minimum value, Maximum value, Number of divisions)
	2nd-axis range (double ymin, double ymax, int ydiv)
19th line	Range against the 2nd-axis direction in the local coordinate system. (Minimum value, Maximum value, Number of divisions)
	3rd-axis range (double zmin, double zmax, int zdiv)
20th line	Range against the 3rd axis direction in the local coordinate system. (Minimum value, Maximum value, Number of divisions)

12-6. Calculate Far Field (JMAG-Studio)

12-6-1. Function

Calculates the electromagnetic field in the region that is sufficiently far from the finite element analysis region, by using the equivalent surface current on the specified surface of analysis model.

12-6-2. Important Notes Regarding the Use of This Function

- This cannot be used in 2D equivalent models.
- This cannot be used for the models with the periodic boundary condition set.

12-6-3. Procedures for Executing the Tool

1. From the menu bar in JMAG-Studio, select [Tools] > [Calculate Far Field].

The [Calculate Far Field] dialog box will appear.

2. Specify the following items.

- [Equivalent Surface Current File]: Specify an equivalent current file (*.srfcur) used for far-field calculation. This file is created in a frequency-response electromagnetic-wave analysis (WV) or transient-response electromagnetic-wave analysis (TD).

  In the WV module, specify the equivalent current calculation surface at the setting conditions and flag the output control item.

  In the TD module, specify the equivalent current calculation surface at the setting conditions.

  This cannot be used for 2D equivalent models or if the model length is not finite in a specific direction because of the periodic boundary conditions.
- [Symmetry Boundary Condition]: Symmetry set here is relative to the equivalent current.

  For example, when the model section is handled as a natural boundary (or perfect magnetic wall) in the electromagnetic wave analysis model, the equivalent current flows parallel to the section, and it is “Symmetrical” in far field calculations. Likewise, when the model section is a symmetry boundary (perfect electric wall), the equivalent current intercepts the section perpendicularly, and it is “Natural” in far field calculations. In any other cases, it is “No.”
- [Radius (m)]: The radius should be the distance to the observation point. Distributions of the electric and magnetic fields on the specified quadratic surface are output.

- [Number of divisions along theta-direction]: Specify the number of divisions in the θ-direction on the quadratic surface to be created.
- [Number of divisions along phi-direction]: Specify the number of divisions in the ϕ-direction on the quadratic surface to be created.
- [Memory Size (MByte)]: Specify the memory size to be used for calculation.

3. Click [Calculate].

12-6-4. Output Items

- Electric field (V/m)
- Magnetic field (A/m)
- Radiated power density (W/m^2)

12-7. Creating Averaged Loss (Power Loss File) (JMAG-Studio)

12-7-1. Function

This tool reads the result files (plot or jplot) that contain the Joule loss data calculated by the magnetic field analysis, and calculates the average of the data over a time period for each element to create the Joule loss file (loss file) for thermal analysis.

If the range of time variation is large in thermal analysis compared to that in the magnetic field analysis, it can use the data of time-averaged loss variation in the magnetic field analysis as the input for the thermal analysis. It is also possible to use as the input the loss in the mover averaged in the circumferential direction. This averages out the loss distribution along the same direction. This is useful when the mover is rotating.

12-7-2. Important Notes Regarding the Use of This Function

- This tool can be used for the transient response analysis (TR, DP) result files.
- If the result file of the magnetic field analysis contains only part of the time or step data to be averaged, it creates the loss file based on the average of the partial data contained. If the result file does not include the specified data to be averaged, it outputs the loss file with value zero(0).

12-7-3. Procedures

1. **From the menu bar in JMAG-Studio, select [Tools] > [Creating Averaged Loss].**

   The [Creating Averaged Loss] dialog box is shown.

2. **Set the magnetic field analysis file to be used as reference data.**

   - [File Name of Magnetic Analysis Result]: Click [Browse] and select the PLOT file or JPLOT file. If selecting the JPLOT file that contains the input data of conditions and materials, you do not need to specify [Magnetic Field Analysis Input File].

- [File Name of Magnetic Analysis Input]: Click [Browse] and select the JCF file or solver data file that corresponds to the PLOT file or JPLOT file selected in [File name of Magnetic Analysis Result].

  MEMO When referencing a file obtained from the analysis where the patch mesh or adaptive mesh feature is used for the region or solid data, specify the file, *_input.jcf, as the input located in the same folder as the magnetic field analysis input file. This file contains the information about the mesh generated by the patch mesh and adaptive mesh features added to the input data used in the magnetic field analysis.

3. Set the Joule loss file for thermal analysis which is generated when this tool completes.

- [File Name of Loss]: Click [Browse] and specify the name of the Joule loss file and where to save the file.

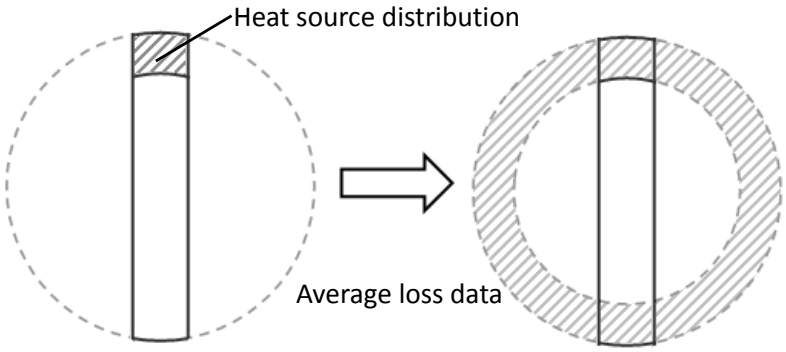
4. Set the range of the loss data that can be referred to in the magnetic field analysis PLOT file specified in step 2.

The data covered by the range specified is averaged for creating the Joule loss file. To set the range, first select from the following a method for specifying the range, then complete the entries for the setting.

- [Step]: Define the range of the loss data to be used by step. Enter the following items:
  - [Start of Step]
  - [End of Step]
- [Time]: Define the range of the loss data to be used by time. Enter the following items:
  - [Start Time (sec)]
  - [End Time (sec)]

MEMO If the time entered does not correspond to any step, the loss value obtained by linearly interpolating the loss between the proximate steps will be used.

5. **For a model with rotation motion, select [Yes] under [Average of Rotation] to average the loss data with respect to the circumferential direction.**
   For a model with translation motion, select [No].



Example: Average the loss data

MEMO If a 2D or an axisymmetric model is used for the analysis result file, the [Average of Rotation] option is not available.

6. **If selecting a solver data file for [File Name of Magnetic Analysis Input] in step 2, specify the memory size to be used for running this tool in [Memory Size (MB)].**

   If selecting the JCF file in the same step, the text does not appear.

7. **Click [Execute].**

12-7-4. Output File

- Loss file (*.loss)

12-8. Temperature for Structural Analysis (JMAG-Studio)

12-8-1. Function

The Temperature for Structural Analysis tool is used to create a static structural analysis file including the temperature data imported from the plot file of a thermal analysis.

Important **Tcl/TK must be installed when using the [Temperature for Structural Analysis] tool. For information about installing Tcl/TK, see the JMAG Installation Manual.**

12-8-2. Procedures

1. From the menu bar in JMAG-Studio, select [Tools] > [Temperature for Structural Analysis].

The [Temperature for Structural Analysis] dialog box is displayed.

2. Specify the following items.

- [Structural Analysis Input]: Specify a JCF file (*.jcf) for a structural analysis or a NASTRAN file (*.dat) and a PLOT file with thermal analysis is added.
- [Thermal Analysis Output]: Specify a PLOT file with thermal analysis to add to a JCF file (*.jcf) or NASTRAN file (*.dat).
- [Structural Analysis Output]: Name the structural analysis result file created after executing this tool. The file is to include the temperature data. The file is either a JCF file (*.jcf) or a NASTRAN file (*.dat).
- [Number of Steps]: Specify the step number of the thermal analysis to which temperature data is to be added.
- [Size of Memory (MB)]: Specify the memory size to be used for calculation.
- [Base of Temperature Flag]: Select the temperature data to export from the list.
  - 1: Export the temperature at each step.
  - 0: Use the temperature at the first step as the reference temperature; export the difference from the reference temperature at the second step and later steps. The temperature at the first step is not exported.
  - -1: Export the temperature at the first step, and then export the cumulative temperature at the second step and later steps.

3. **Click [Set].**

   The settings are confirmed.

4. **Click [Execute].**

12-8-3. Output Items

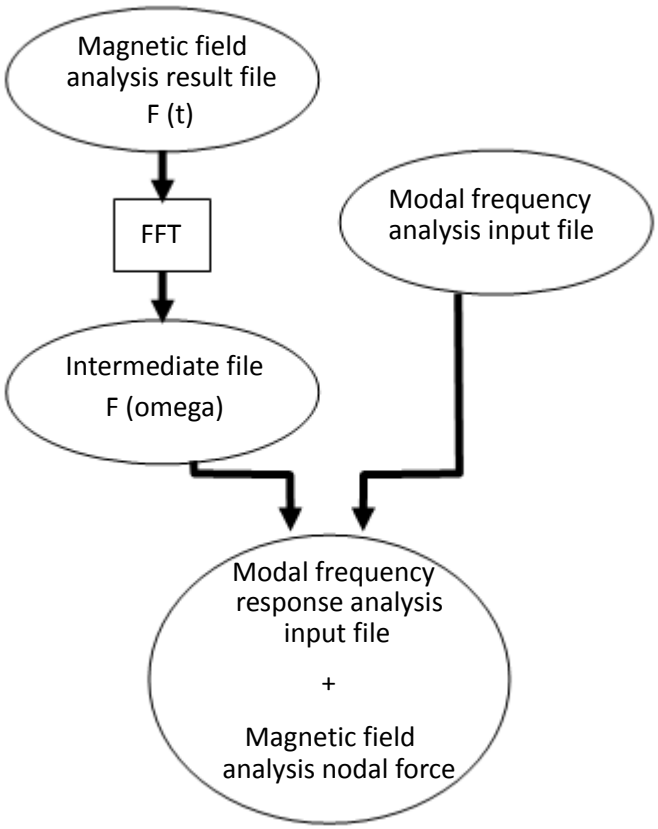
- Temperature (Celsius)

12-9. Force Mapping for Structural Analysis

12-9-1. Function

The electromagnetic force included in a result file (*.jplot, *.plot) for magnetic field analysis is mapped to the input file (*.jcf, *.dat) in a structural analysis.

The load for frequency decomposition by fast Fourier transformation is mapped for the input file of a structural analysis (frequency response analysis).


Magnetic field analysis result file F (t)
FFT
Intermediate file F (omega)
Modal frequency analysis input file
Modal frequency response analysis input file
+
Magnetic field analysis nodal force

Basic concept for the mapping of electromagnetic force

The JCF file of structural analysis can be converted to the NASTRAN file, and vice versa.

12-9-2. Important Notes Regarding the Use of This Function

■ Input File of Structural Analysis

- A JCF file (*.jcf) and a NASTRAN file (*.dat) can be specified as an input file for a structural analysis.
- The electromagnetic force is outputted to the following card in a NASTRAN file, when specifying a NASTRAN file as a structural input file.
  - Static analysis (SOL = 101)
    An electromagnetic force is outputted as a static load for FORCE (nodal)
  - Frequency response analysis (SOL = 111)
    An electromagnetic force is outputted to RLOAD1 (frequency dependent load)
- Use the JCF file for a structural analysis that is exported from JMAG-Designer when starting the force mapping tool for a structural analysis model from JMAG-Designer. The JCF file exported from a structural analysis from JMAG-Studio when starting from JMAG-Studio can also be used.
- An electromagnetic force can be mapped on an input file for a 3D structural analysis using a 2D result file of a magnetic field analysis. In this case, the map area of an electromagnetic force in a structural analysis model needs to exist in +Z direction.

■ Reference file for magnetic field analysis

- A JPLOT file (*jplot) or a PLOT file (*.plot) can be specified as a reference file for a magnetic field analysis.
- Specify a JCF file when a PLOT file is specified as a reference file for a magnetic field analysis.
  - A JCF file for a magnetic field analysis (static analysis, transient response analysis, frequency response analysis) can be specified using the force mapping tool in a structural analysis file in JMAG-Designer. However, a JCF file for a magnetic field analysis (static analysis) cannot be specified, when the input file for the structural analysis is a frequency response analysis file.
  - A JCF file for a magnetic field analysis (transient response analysis, frequency response analysis) can be specified using the force mapping tool in a structural analysis file in JMAG-Studio.
- The magnetic field analysis data that includes the axial reverse periodic boundary cannot be used with this tool.
- Multiple result files for a magnetic field analysis can be specified using a force mapping tool in a structural analysis file in JMAG-Designer.

12-9-3. Procedures for JMAG-Designer

1. **Select [Tools] > [Force Mapping for Structural Analysis] from the menu bar.**

   The [Force for Structural Analysis] dialog box is displayed.

2. **Click [Browse], and then select the structural analysis JCF file (*.jcf), or the NASTRAN file (*.dat), for mapping electromagnetic force.**

   If the electromagnetic force condition is set in the selected file, a message dialog box, "Is the electromagnetic force condition of the input file reflected?" is displayed. If Yes is selected, the settings of the electromagnetic force condition are applied.

3. **Set the following items for the structural analysis data specified in step 2.**

   - [Target]: Select materials that are to be the target of mapping.
   - [Reference Coordinate System]: Specify this if the magnetic field analysis to reference uses a 2D model and the structural analysis uses a 3D model. The electromagnetic force is mapped for the structural analysis model by extruding the magnetic field model in the Z-axis direction of the specified coordinate system.

     Click [Set]. The [Coordinate System] dialog box is displayed. Specify the origin, a point on the X-axis, and a point on the XY-plane, and then click [OK].

4. **Specify a value in the [Basic Frequency] box in [Magnetic Field Analysis Model], when a structural analysis (frequency response analysis) file is specified in [Structural Analysis Model].**

   The frequency for a magnetic field analysis result needs to be multiply the value of a basic frequency.

5. **Click the [Add] button to display the [Edit] dialog box in [Magnetic Field Analysis Model], and then specify and set related settings for a result file of a magnetic field analysis.**

   Specify the following items in the [Edit] dialog box.

   - [Output File]:

     Specifies a JPLOT file or a PLOT for a magnetic field analysis.
   - [Input File]:

     Specify an input file (*jcf) for a magnetic field analysis, when specifying a PLOT in [Output File]. Analysis types that are different in [Output File] and [Input File] can also be specified (for example transient response analysis and frequency response analysis). However, an input file for a magnetic field analysis (static analysis) cannot be used, when the structural analysis file (frequency response analysis) selected in step 2 is specified.

Reference

For more information about referencing magnetic field analysis files, see "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction" on page 801.

- [Reference Interval]:

  Specifies which electromagnetic force for a step to map from a magnetic field analysis result.

  Specify a step to be referred from the result file types of a magnetic field analysis in [Start Step], when specifying the structural analysis (static analysis) file selected in step 2.

  The items that can be specified differ, based on the types of result file in a magnetic field analysis, when specifying the structural analysis (frequency response analysis) file selected in step 2

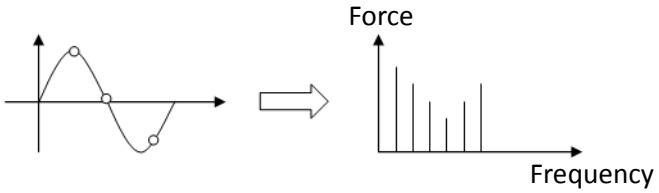
  - Specify a step ID for [Start Step] and [End Step] to set an area for reference, when referring to a result file of a magnetic field analysis (transient response analysis).
  - Specify a step ID in [Start Step] and set a step to be referred, when referring to a result file for a magnetic field analysis (frequency response analysis).

  MEMO A result file for a magnetic field analysis (static analysis) cannot be referred for an output file, when a structural analysis (frequency response analysis) is specified as an input file.

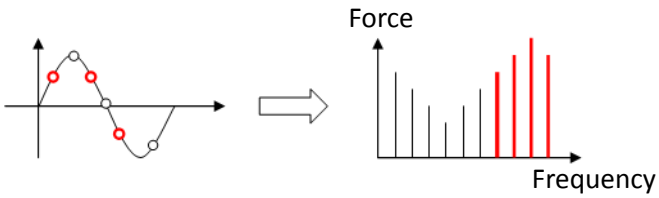
- [FFT Resolution]:

  The time resolution in a fast Fourier transformation can be specified, when the structural analysis (frequency response analysis) file is specified in step 2.

  Specify a value larger than one when mapping the electromagnetic force in high frequency from a magnetic field analysis result in a structural analysis model.



[FFT Resolution] is '1'



[FFT Resolution] is larger than '1'

- [Scaling Factor]:

  Specifies the multiplication for mapping the electromagnetic force in a magnetic field analysis.

- [Extend Distance [mm]]:

  This setting is specified when referencing a result file for a magnetic field analysis in 2D, and an input file for the structural analysis in 3D.

  The result for a magnetic field analysis in 2D is expanded in Z-axis direction by the values set for extended value. This is used for mapping a result file in a structural analysis in 3D.

  The result for a magnetic field analysis in 2D is expanded in Z-axis direction by the values set for extended value. This is used for mapping a result file in a structural analysis in 3D.

6. Specify each item in the [Edit] dialog box, and then click [OK].

The [Force for Structural Analysis] dialog box is displayed.

The result file in a magnetic field analysis is added to the [Magnetic Field Analysis Model] table.

7. Repeat steps 5 and 6 to specify multiple result files for a magnetic field analysis as required.

Select a file from the [Magnetic Field Analysis Model] table in the [Force for Structural Analysis] dialog box, and then click [Edit] when modifying the settings in a result file for an added magnetic field analysis.

Select a file from the [Magnetic Field Analysis Model] table, and then click [Delete] when deleting an result file of a magnetic field analysis.

MEMO The settings can be edited by double-clicking the [Magnetic Field Analysis Model] table or selecting [Edit] from the right-click menu.

MEMO A file can be deleted by selecting a file from the [Magnetic Field Analysis Model] table, and then pressing the [Delete] key on the keyboard, or selecting [Delete] from the right-click menu.

8. To display the progress of mapping, select the [Show Status] check box.

9. Click [Run].

The [Save As] dialog box is displayed.

10. Specify the location to save the JCF file or NASTRAN file, and then click [Save].

The mapping stars.

If a JCF file is used, the JCF file is exported. If a NASTRAN file is used, the NASTRAN file is exported.

12-9-4. Procedures for JMAG-Studio

1. **Select [Tools] > [Force for Structural Analysis] from the menu bar.**

   The [Force for Structural Analysis] dialog box is displayed.

2. **Click [Browse] next to the [Structural Input File] box, and then select the structural analysis JCF file (*.jcf), or the NASTRAN file (*.dat), for mapping electromagnetic force.**

   If the electromagnetic force condition is set for the selected file, a message dialog box, “Is the electromagnetic force condition of the input file reflected?” is displayed. If [Yes] is selected, the settings of the electromagnetic force condition are reflected.

   To use a solver data file of structural analysis exported from JMAG-Studio, specify the file as a NASTRAN file.

   For example, if a solver data file “AAA” is to be used, add the NASTRAN file extension (*.dat) to change the file to “AAA.dat”. When specifying the structural input file for this tool, select [Nastran files (*.dat)] in the [Files of type] combo box, and then select “AAA.dat”.

   MEMO JMAG-Studio automatically starts when a NASTRAN file is specified.

3. **Set the following items for the structural analysis data specified in step 2.**

   - [Use existing Set Target]: Map to the target specified with the electromagnetic condition set for the JCF file of structural analysis.
   - [Target]: Select a check on the materials in the list that are to be the object of mapping.
   - [Scaling Factor]: The scale factor should be determined with respect to the thickness of a board in 2D analysis.

     Example: If a thin board has a thickness of 1 mm in 2D analysis, enter “0.001” as the scale factor. If 3D analysis must be dealt with, enter “1.0”.

4. **Set the following items for the magnetic field analysis data to reference.**

   - [Input File]: Set a solver data file for the magnetic field analysis (transient response analysis or frequency analysis) or a JCF file.
   - [Output File]: Set the plot file or jplot file containing the analysis results of the file specified for [Input File].

     ▶▶ Reference

     For more information about the magnetic field analysis file to be referenced, see P.801 "10-6. Coupling Magnetic Field Analysis and Structural Analysis in One Direction".

- [Start Step] / [End Step] of [Reference Interval]: Set the range, in steps, to be referenced from the magnetic field analysis file. Note that when a static analysis file is set in [Structural Input File], the [End Step] cannot be set.
- [FFT Resolution]: Specify the time resolution of the fast Fourier transformation (FFT) for frequency response-based structural analysis. If a number larger than 1 is entered, the time resolution of the FFT will be increased over the normal.

  This item cannot be set when a static analysis file is set in [Structural Input File]. Likewise, this item cannot be set when a frequency response analysis file is set in [Input File]/[Output File] of the [Magnetic Field Analysis Model] group box.
- [Convert Full Model]: Select this check box when the magnetic field analysis model to be referenced is a partial model and a full model expansion is desired.

  The rotational axis when conducting a full model expansion is the Z-axis of the global coordinate system.

5. If the mesh shape is different between the magnetic field analysis data and the structural analysis data, select the [Use Different Mesh between Magnetic Field and Structural].

6. Set mapping parameters if [Use Different Mesh between Magnetic Field and Structural] is selected.

- [Map from 2D (Magnetic Field) to 3D (Structural)]: If it is required to expand the reference data of magnetic field analysis to 3D, select the check box and set the following parameters.
  - [Extend Distance (mm)]

  When the data of a magnetic field to be referred to is based on a 2D model while the structural analysis project to be executed must be based on 3D data, if this function is selected, the 2D data of the magnetic field analysis will be expanded to 3D data, thus the user can use the expanded data for their 3D structural analysis.

  The extended axis for the 3D expansion is the Z-axis of the global coordinate system.

7. To display the progress of the mapping, select the [Show Status] check box.

8. To conduct a file conversion alone without mapping, select the [Do Not Map Force] check box.

9. Click [Run] and the [Save] dialog box will open.

10.Set a file name and the file type.

For the file type, a JCF file or a NASTRAN file can be selected.

11.Click [Save].

The mapping or conversion will be run.

If the [Show Status] check box is selected, the [Show Status] dialog box will be displayed.

- In case calculation is made by using the Patch Mesh function or Adaptive Mesh function in the region data or solid data for version 8.2 or earlier, mesh information is not included in the JCF File or solid file that are created. Therefore, such files could not be referenced in this function.
- In case the same calculation is executed with version 8.3 or later, a file called "*.input.jcf" will be output after the calculation is completed. This file is made by inputting the mesh information to the JCF File that executed the calculation. It is possible to specify this "*.input.jcf" as Magnetic Field Analysis Input File.
  * In the "*" of the "*.input.jcf" the name of the JCF File that executed the calculation will be entered.

12-9-5. Output File

- JCF file (*.jcf) of the structural analysis or a NASTRAN file (*.dat)

12-9-6. Executing the Tool from the Command Line

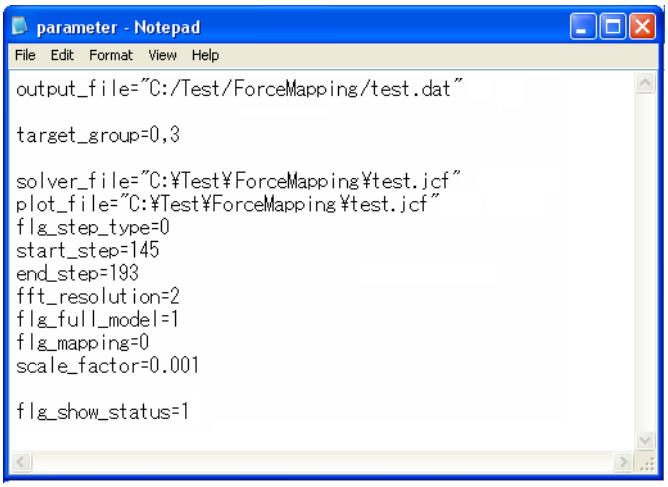
The Force for Structural Analysis tool can be executed using the command line. Using the command line is useful for mapping multiple files.

1. Create a parameter file (*.txt).

Write the change for parameters in the input data created in step 1.

The following should be noted.

- The text format (*.txt) should be used.
- The name of the file can be any name.
- One parameter file needs to be created for one calculation.
- Parameter name = Value



▶▶ Reference

For more information about the parameter name, see "■Parameters" on page 964.

2. Start the command prompt, and enter commands. Alternatively, crate a batch file (*.bat) containing commands, and then execute the commands.

Enter the following commands.

■ Specifying a JCF file as an input file

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\structure\mod\ForceMappingGui.exe
-file="name and path of the parameter file (*.txt)"
-jcf="name and path of the JCF file (*.jcf) of structural
analysis"
```

Example

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\structure\mod\ForceMappingGui.exe
-file="D:\JMAGData\parameter.txt"
-jcf="D:\JMAGData\FEM\Test.jcf"
```

■ Specifying a NASTRAN file as an input file

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\structure\mod\ForceMappingGui.exe
-file="name and path of the parameter file(*.txt)"
-nas="name and path of the Nastran file(*.dat) of structural
analysis"
```

Example

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\structure\mod\ForceMappingGui.exe
-file="D:\JMAGData\parameter.txt"
-nas="D:\JMAGData\FEM\Test.dat"
```

Important **A command and an argument should be defined in one line. To use two lines, enter "^" (hat) at the end of the line.**

MEMO To use a batch file, enter either of aforementioned command in the text editor, specify a name, and then save. Change the extension of the file to (*.bat), and then double-click the file, the command is executed.

When the command is executed, the File for Structural Analysis tool is executed.

■ Parameters

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description	Studio/ Designer
target_group	id		Material ID to map	Both
scale_factor	Double		Scale factor	Studio
coordinate_system	Double		Define the center point (x,y,z), a point on the X-axis (x,y,z), and a point on the XY-plane by separating them with a comma.	Designer
solver_file	String		Magnetic field analysis file to reference	Both
plot_file	String		Plot file or jplot file	Both
flg_step_type	Flag	0 1	Type Specify by step Specify by frequency	Both
start_step	Int		Start step	Both
end_step	Int		End step	Both
frequency	Double		Frequency	Both
fft_resolution	Double		FFT resolution	Both
flg_full_model	Flag	0 1	Convert to full model Do not convert Convert	Studio

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description	Studio/ Designer
flg_mapping	Flag	0 1	Use the same mesh between magnetic field analysis and structural analysis Use different mesh between magnetic field analysis and structural analysis	Studio
flg_extension	Flag	0 1	Map from 2D magnetic field analysis to 3D structural analysis Do not map Map	Studio
distance_extension	Double		Extended distance (mm)	Studio
flg_mapping_force	Flag	0 1	Map electromagnetic force Map Do not map	Studio
flg_show_status	Flag	0 1	Flag for displaying the status Display Do not display	Both

12-10. Force Export for Virtual.Lab (JMAG-Designer)

The nodal force exported from a magnetic field analysis (transient analysis) from JMAG-Designer can be used for the vibration/noise analysis in Virtual.Lab which is the general purpose electromagnetic optics design and evaluation program when using the electromagnetic force tool in Virtual.Lab.

This tool imports the results file (*.jplot) for JMAG-Designer that includes results of electromagnetic force and extracts information of electromagnetic force and surface mesh. The informations exported in Universal file format which can be imported into Virtual.Lab.

This tool has the following functions.

- Extracting information on surface mesh and exporting to a universal file
- Exports electromagnetic force to a Universal file.
- Expands results for a 2D model in JMAG-Designer to a 3D model.
- Results for a partial model in JMAG are expanded to a full model based on the information for a periodic boundary condition

12-10-1. Data Transferred Between Programs and where it is Received

- JMAG magnetic field analysis (transient response analysis) electromagnetic force >> Virtual.Lab vibration/noise analysis.

12-10-2. Flow of Analysis Linked with Virtual.Lab

1. **Run a magnetic field analysis (transient response analysis) in JMAG-Designer and calculate the nodal force.**

   The torque (nodal force) condition needs to be set when calculating nodal force.

2. **Start the force export tool for Virtual.Lab and import the result file (*.jplot) from step 1.**
3. **Select the part to export to a Universal file and run the export function.**
4. **Run a vibration/noise analysis in Virtual.Lab using the Universal file in step 3..**

12-10-3. Important Notes Regarding the Force Export for Virtual.Lab

- This tool is only supported for result files of a magnetic field analysis (transient response analysis).
- The magnetic field analysis (static analysis, frequency analysis) is not supported.
- This tool is only supported for nodal force. The Lorentz force is not supported.
- This tool is not supported for the following models.
  - Partial model with axisymmetric
  - Partial model with translation periodicity
- The Universal file exported from this tool can be used in Virtual Lab Rev 11. or later.

12-10-4. Running procedures for the Force Export for Virtual.Lab

1. **Select [Tool] > [Force Export for Virtual.Lab] from the JMAG-Designer menu bar.**

   The [Force Export for Virtual.Lab] dialog box is displayed.

2. **Select [Browse] in [Input File], and specify a result file (*.jplot) for a magnetic field analysis for JMAG.**

3. **Select [Browse] in [Output File], and specify the path for the Universal file (*.unv) that is exported.**

4. **Select the check box for the part to export to a file in [Parts To Export].**

   To select all the parts, select [Check All] and to clear the selection for all the parts, select the [Check None] button.

   The settings are automatically specified for parts set with the torque (nodal force) condition during calculation in JMAG-Designer.

5. **Specifies a conversion option for a model.**

   - [Convert to 3D]: Specify this setting and set the values in the [Length] and [Divisions] box to expand a result of a magnetic field analysis for a 2D model to a 3D model.

     Specify the thickness of a model in the [Length] box. The unit is (m). The value specified in the [Stack Length] box for the full model conversion in JMAG-Designer is the default value.

     Specify the number of divisions in the [Divisions] text box for models that are divided in multiple axial direction.

- [Convert to Full Model]: Specify this setting when expanding a partial model from a result data to a full model based on the settings for a periodic boundary condition.

6. Specify the range of steps to export to a file.

Select the specifying method and specify the required items.

- [All Steps]: Export the results for the entire steps.
- [Steps from End Steps]: The result values after the specified value that is subtracted from the number of the last step are exported.

  For example, if the value of the last step is [10] and the [Steps] is set at [3], then the results for steps after [8] is exported to a Universal file.
- [Specify Step Range]: Specify [Start Step] and [End Step] and export the entire result steps in the range.

7. Specify whether or not to extend the result data based on the time axis.

- [No expansion]: Select this setting when the result data is not extended.
- [Expand from Half Cycle]: This setting is specified when the result has time series periodicity and exported with a half period result.

8. Select the [Quit on Export] check box when automatically ending the tool when exporting is finished.

9. Click the [Export] button.

The Universal file is exported.

12-10-5. Export Files

- Universal file (*.uni)
- JMAG Binary Plot File (*.jplot)

12-11. Exporting Multi-purpose Files (JMAG-Designer)

12-11-1. Function

Extracts result data from a JMAG-result file (*.jplot) and converts data to Nastran bulk data format to export. This function does not simply export files, where merging result data from multiple JPLOT files and adding features such average or fast Fourier transform (FFT) can also be set.

The exported Nastran files can be used in other simulation software other than JMAG.

12-11-2. Data that can be Exported

The following types of data can be exported.

- Mesh *Except air region
- Step information (analysis time or analysis frequency)
- Result data (physical quantities)
  The following export format can be selected.
  - Value of each analysis time or analysis frequency
  - Average of each step based on time series, or by adding the entire values from each frequency into a value of a single step
  - The value of frequencies is broken down using fast Fourier transform (FFT)

12-11-3. Important Notes Regarding the Use of This Function

- Nastran bulk data files of Ascii format are standard when exporting file format, in which some parts use format which are specifically for JMAG.
- Result data of a partial model can be converted into a full model. In this case, the mesh model is generated by specifying a mirror copy or rotation copy to a partial model.
  However, the conversion is based on either the periodic boundary condition or partial model condition settings. Conversion specified with settings from symmetry boundary condition or natural boundary condition is not supported.
- 2D models or axisymmetric models can be expanded to 3D models for export.
- The air region created using the function to generate mesh automatically (standard, slide) is not outputted.
- Result data is outputted in SI units.

- Result data (nodal force or magnetic flux density) to output or parts can be selected. Selecting and adding multiple output data can also be outputted. For example, loss (heat generation) is outputted by adding hysteresis loss and joule loss.
- Result data extracted from multiple JMAG result files with analysis types that are the same or different can be exported in a single file. The following are examples.
  - Centrifugal force for the nodal force in a magnetic field analysis (transient response analysis), and structural analysis (static analysis).
  - Total value of Joule loss for each frequency in a magnetic field analysis (frequency response analysis).
  - Total value of averaged joule loss for a magnetic field analysis (transient response analysis) and loss in an iron loss analysis.

  The scaling setting in a result data or offset is supported.

12-11-4. Running Procedures

1. **Select [Tool] -> [Multi-Purpose File Export] from the JMAG-Designer menu bar.**

   The [Multi-Purpose File Export Tool] dialog box is displayed.

2. **Select [Browse] in [Output File] to display the [Target File] dialog box, and then specify the name and path for the Nastran file (*.nas) that is exported.**

   The [Add] button in [Reference Result Files] is active when [Output File] is specified.

3. **Select an export format.**

   The following export format can be selected.

   - [Static Analysis]

     Outputs value for a single analysis step (exports files corresponding to static analysis)

   - [Frequency Analysis]

     Outputs value for area in frequency (exports files corresponding to frequency response analysis).

The selected export format may not be supported, based on the analysis type for the JMAG result file specified in step 4. Confirm the status in the following table.

Table of analysis type for JMAG result file and export format

		Static analysis	Frequency analysis
Magnetic field analysis	Static analysis	O	X
	Transient analysis	△*1	△*2
	Frequency analysis	△*3	
Thermal analysis	Steady state analysis	O	X
	Transient state analysis	△*1	△*2
Structural analysis	Static analysis	O	X
	Frequency analysis	△*3	O
	Eigenmode analysis	△*3	O
Electric field analysis	Static analysis	O	X
	Frequency analysis	△*3	O
	Current distribution analysis	O	X

*1 The entire steps or specified steps in a section is averaged, or only a single step in an output step is specified.

*2 Fast Fourier transformation (FFT) is run.

*3 Values are outputted when the total value and entire frequency exist.

MEMO For more information about result files for a transformer analysis study, see the table above for a magnetic field analysis.

Important **Confirm the settings set in each result file for JMAG, when adding a result file for JMAG and modifying the export format.**

4. Specify the JMAG result file for extracting data and the type of data extraction.

Click [Add] in [Reference Result Files], to display the [Select Input Result Files] dialog box. Some sections for the settings in this dialog box are different based on the export format selected in step 3.

- [Result File]

  Click the [Browse] button to open the [Open file dialog] dialog box, and then specify a result file for JMAG (*.jplot).

- [Part] table

  The parts that are included in the specified result file for JMAG, are displayed in the [Part] table. Select the check box of the part to be exported to a file.

  MEMO The entire check boxes for parts can be switched when using each button in [Select All] and [Deselect All].

- [Output] table

  A list of result data is displayed in the [Output] table, when specifying a result file for JMAG. Select the check box of the result data to be exported to a file.

  Specifying [Factor] and [Offset] in each result data is optional.

  Important **Result data included in a result file for JMAG is exported in values of SI units. Specify a value for [Offset] in SI units.**

  MEMO The entire check boxes for result data can be switched when using each button in [Select All] and [Deselect All].

- [Option]

  Specifies the output area for a result data or the option to add features. The options that can be specified differ based on the export format selected in step 3.

  - [Target Step]

    Specify whether the entire step is outputted or a specified section is outputted, when multiple result data for an analysis step is outputted in a result file for JMAG.

    A single step needs to be outputted when [Static Analysis] is specified in [Output Type]. Therefore the step is averaged automatically in a time axis direction. The section that is averaged is specified in [Target Step]. A value for the step that is outputted when specifying the same step ID in [Start] and [End].

- [Periodicity]

  This setting can be specified when [Frequency Analysis] is set in [Output Type], and the analysis type in a result file for JMAG is for a transient response analysis. Select [None] when there is no periodicity in the section specified in [Target Step] or [1/2] when there is half periodicity.

  Select the [Set Harmonic Order] check box and set the harmonic order in the box, when specifying the order of the frequency that is outputted.

- [Extended to the Full Model the Partial Model]

  A partial model can be converted to a full model when the result file for JMAG is for a partial model.

  Select [Partial Model Refers to a Condition], when referring to settings in a partial model condition.

  Select [Manually] when manually setting the conversion method. Select the [Extended Period with Reference to the Periodic Boundary] check box when referring to settings for the periodic boundary condition to convert to a full model. Select each check box in [Expanded in the XY Plane Mirror], [Expanded in the YZ Plane Mirror], and [Expanded in the ZX Plane Mirror], for models with symmetry between the XY, YZ, and ZX planes.

  MEMO The partial model condition can be set on a magnetic field analysis and thermal analysis. Rotational periodicity, translational periodicity, or symmetry boundary (mirror) can be set on a partial model condition.

- [Extended to 3D Model]

  Result files for JMAG that are 2D models or axisymmetric models, can be expanded to a 3D model.

  Specify a distance extending in a normal direction in the [Height] box for 2D models. Select a unit to apply to the height in [Unit] as required.

  The area around the axis is expanded 360 degrees for an axisymmetric model.

- [Coordinate System]

  The result data is summarized in a mesh model (hear on after referred to as 'output mesh model') that is included in a JMAG result file selected in the [Output Model] list of the same dialog box. This can be done when multiple result files are added in [Reference Result Files] of the [Multi-Purpose File Export] dialog box.

  The positions are adjusted by using the coordinate system when positions differ for the mesh model (hear on after referred to as 'selected mesh model') included in a result file selected in the [Select Input Result Files] dialog box, and an output mesh model.

  - Specify a direction vector in the coordinate system for an output mesh model in [Primary Axis]. This vector is equivalent to the X-axis direction for a selected mesh model.
  - Specify the direction vector in the coordinate system for an output mesh model in [Secondary Axis]. This vector is equivalent to the Y-axis direction for a selected mesh model.
  - Specify a position in the coordinate system for an output mesh model in [Origin]. This vector is equivalent to the orgin coordinates for a selected mesh model. Select a unit to use for the coordinate value in [Unit] as required.

Click [OK] when the settings are complete to display the [Multi-Purpose File Export Tool] dialog box.

5. Repeat the operations in step 3 to add files in [Reference Result Files], when extracting data from multiple result files for JMAG to export in a single Nastran file.

Select [Reference Result Files], and then click the [Edit] button when modifying the settings in a result file for JMAG that is added in [Reference Result Files].

Click the [Remove] button, and then select [Reference Result Files] when deleting result files for JMAG that is added in [Reference Result Files].

6. Select an output mesh model from the [Output Model] list.

JMAG result files added in [Reference Result Files] are displayed in the [Output Model] list. Select a file with an output mesh model from the list.

7. **Specify a value in [Number of Mesh Extended 3D Model] to create a mesh model, when expanding a mesh to 3D for a 2D model or an axisymmetric model.**

   Specify the number of divisions for the direction in height when expanding mesh for a 2D model.

   Specify the number of divisions for a circumferential direction (360 degrees), when expanding mesh for an axisymmetric model.

8. **Confirm the contents exported in the Nastran file in [Output Item].**

   The result data exported to a Nastran file is displayed in hierarchy in [Output Item].

   Confirm the settings and modify the settings as necessary.

9. **Select [Export] to export the data into a Nastran file.**

   The Nastran file with the name specified in step 2 is exported.

10. **Select [Save Setting], and then specify a file name and directory to save the file when saving the settings to a file.**

    The settings file for the tool to export multiple files can be imported by clicking the [Open Setting] button in the tool for exporting multi-purpose files (xml).

    This file is used for running command lines that are explained in a later section.

12-11-5. Export Files

- Ascii Nastran file (*.nas)
  - The mesh data is exported in compliance with Ascii Nastran file format.
  - Registered keywords are outputted in compliance with Nastran format. For example, the electromagnetic force is exported as 'FORCE' and 'DLOAD,' and loss (heat source) is exported as 'QVOL' and 'TLOAD.'

12-11-6. Running from the Command Line

The export tool for multi-purpose files can be run from the command line. This function is useful for exporting multiple files.

1. **Start up the tool for exporting multi-purpose files, and then set each item to export the settings file (*.xml) by clicking [Save Setting].**

   For more information about setting methods for the tool to export multi-purpose files, see "12-11-4. Running Procedures" on page 970.

2. **Start up the command prompt to run the specific commands that are described.**
   Create also a batch file (*.bat) with commands, and then execute the batch file.

   The commands described below can be used.

   ```
   "C\Program Files\JMAG-Designer12.0\GeneralInterface.exe"
   -in "settings file (*.xml) path"
   ```

   Important **The command and the argument defined, needs to be described in one line. Insert a caret symbol '^' when describing the command in two lines.**

   MEMO Specify and save the command above in text editor, and then save the file. Modify the extension of the file to '*.bat', and then double-click the file to run the command.

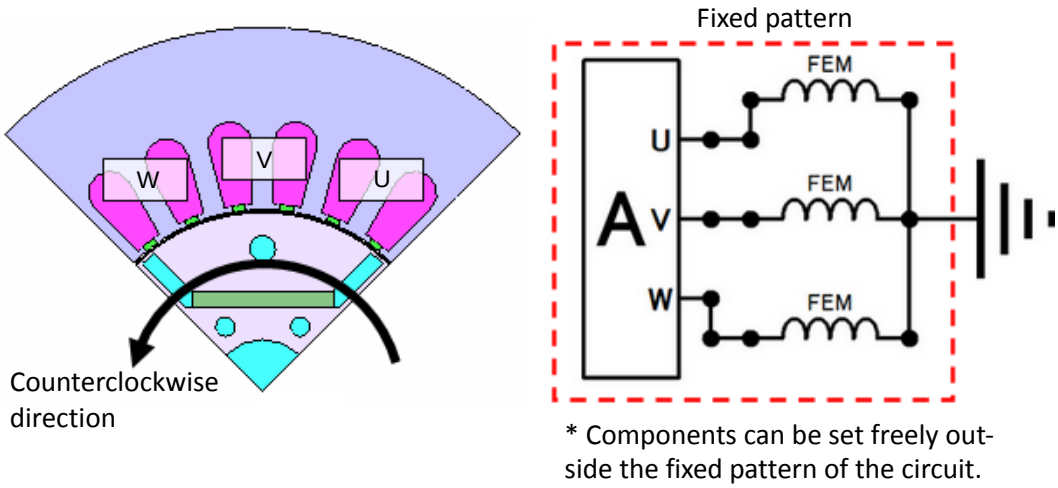
   The tool to export multi-purpose files is run when executing the commands.

12-12. Calculate Motor Inductance (For Three-phase Permanent Magnet Synchronous Motor)

12-12-1. Function

The Calculate Motor Inductance tool is used to obtain the electric current and electric current phase dependency of the d-axis and q-axis inductance in a three-phase permanent magnet synchronous motor using the magnetic flux of the coils.

The Calculate Motor Inductance tool exports a file used for the PSIM Motor Saturation Model.



Example: Mesh drawing and circuit drawing of three-phase permanent magnet synchronous motor

MEMO The calculate motor inductance tool can be run using a third-party job management system.

12-12-2. Flow of Calculation

The inductance of the motor is calculated in the following steps.

1. **Create an FEM model of the magnetic field analysis.**

   Set material properties and conditions, create a circuit, and then generate a mesh.

2. **Start the Calculate Motor Inductance tool, and then import the FEM model created at step 1.**

3. **Specify the parameters for the inductance calculation.**

4. **Run the calculation.**

   Hereinafter, the procedures after step 2 are described.

12-12-3. Files Required to Use the Tool

- JCF file (*.jcf) of the transient response magnetic field analysis (TR, DP).

  MEMO The solver data file cannot be used.

12-12-4. Important Notes Regarding the Use of This Function

The following should be noted when using this tool:

- The following motors can be used with this tool.
  - 3-phase PM synchronous motor
  - Field winding type synchronous motor
- This tool cannot be used with the following motor.
  - Linear motor
- The following conditions must be set for the motors used for this tool.
  - Motion conditions
    The rotation axis and mover must be specified.
  - Three-phase current source component of the circuit
    This tool cannot be used for the motors that have multiple three-phase current source components in the circuit. The three-phase current source component must be used instead of the three-phase power supply (current) of a macro component.
- Each phase of the coils needs to be linked to each FEM coil component.
- The fixed pattern (the area surrounded by a frame in the above circuit diagram) must be used for the arrangement of the circuit components.

12-12-5. Procedures

1. **Select [Tools] > [Calculate Motor Inductance] from the menu bar.**

   The [Calculate PM Motor Inductance] dialog box is displayed.

2. **Click [Browse] for [Input File], and then specify the JCF file that contains the FEM model to calculate.**

3. **Select the current definition.**

   The following are the two types of current specifying methods.

   - [Amplitude/Phase]
   - [d-axis Current/q-axis Current]

4. **Specify the following items.**

   - [Current Frequency (Hz)]
   - [Number of Motor Poles]: Enter the even natural number.

5. **When [Amplitude/Phase] is selected for the current definition, specify the following items.**

   - [Amplitude]
     - [Max (A)]
     - [Min (A)]
     - [Number of Divisions]: Enter the number of divisions from maximum to minimum value of the amplitude. When the same value is entered in [Max (A)] and [Min (A)], [Number of Divisions] will be automatically set to "1."
   - [Current Phase]: The zero point of current phase is determined based on the no-load induced voltage waveform.
     - [Max (deg)]: The maximum value is 360 degrees.
     - [Min (deg)]
     - [Number of Divisions]: Enter the number of divisions from maximum to minimum value of the phase. When the same value is entered in [Max (A)] and [Min (A)], [Number of Divisions] will be automatically set to "1."

6. When [d-axis Current/q-axis Current] is selected for the electric current definition, specify the following items.

- [d-axis Current]
  - [Max (A)]
  - [Min (A)]
  - [Number of Divisions]: Enter the number of divisions from maximum to minimum value of the d-axis current. When the same value is entered in [Max (A)] and [Min (A)], the [Number of divisions] will be automatically set to "1."
- [q-axis Current]
  - [Max (A)]
  - [Min (A)]
  - [Number of Divisions]: Enter the number of divisions from maximum to minimum value of the q-axis current. When the same value is entered in [Max (A)] and [Min (A)], the [Number of Divisions] will be automatically set to "1."

7. To export a "Look up table" file for PSIM, select the [Output PSIM LUT File] check box.

By selecting this check box, two files named "Lb.tbl" and "Lq.tbl" are exported after the analysis is complete. These files can be used as the nonlinear table (Lookup Table) of Ld and Lq in the saturation model of a permanent magnet synchronous motor (Synchronous Machine with Saturation) of the Motor Drive Module for the circuit simulator PSIM.

This check box can be left OFF when it is not used in PSIM.

8. To distribute the inductance calculation, select the [Distributed Calculation (Remote)] check box, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

This is the feature that automatically distributes the calculation to the CPU nodes on the JMAG Remote System network.

For the inductance calculation which has multiple cases due to the variation of current, each case of the calculation is distributed to the CPU nodes.

9. Select the [Save File Magnetic Analysis Result] check box to save the magnetic analysis plot file that is exported during the process of inductance calculation after the completion of the calculation.

The solver report (*.html) is also exported with a plot file.

10. Click [Run].

The [Save As] dialog box is displayed.

11.Specify the directory for running the analysis, and then enter a name for the result file.

12.Click [OK].

The calculation begins and the command prompt starts up to indicate the progress of the calculation process.

12-12-6. Output Items

- d-axis current (A)
- q-axis current (A)
- d-axis inductance (H)
- q-axis inductance (H)

■ Definition of inductance

With this tool, the inductance of the d-axis and q-axis is calculated using the following equations.

$$L_q = \frac{R \cdot I_d - V_d}{\omega \cdot I_q}$$

$$L_d = \frac{1}{\omega} \cdot \frac{dV_q}{dI_d}$$

R: Coil resistance, I_d: d-axis current, V_q: d-axis voltage, I_d: q-axis current, V_q: q-axis voltage, ω: Angular velocity

MEMO For reference, see the following book.
J.R.Hendershot Jr. and Tje Miller. Design of Brushless Permanent-Magnet Motors. Oxford Magna Physics, 1994

12-12-7. Output Files

- d-q Axes inductance calculation result file (*.csv)
- Nonlinear table of d-axis for the Motor Saturation Model “Ld.tbl”
- Nonlinear table of q-axis for the Motor Saturation Model “Lq.tbl”

12-12-8. Executing the Tool from the Command Line

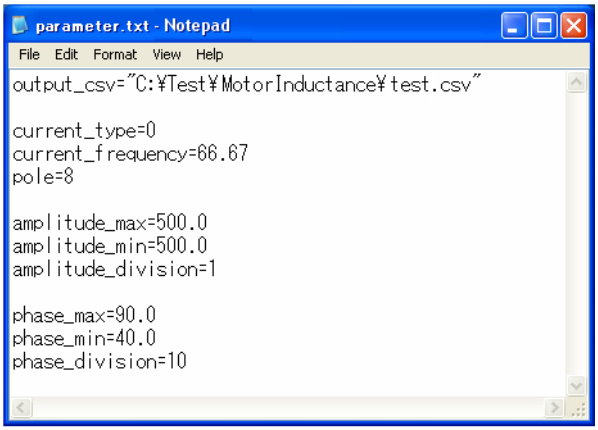
The Calculate PM Motor Inductance tool can be executed using command line. The command line is useful for executing multiple files at once.

1. Create a parameter file (*.txt).

Write the change for parameters of the input data created in step 1.

The following should be noted.

- The text format (*.txt) should be used.
- The name of the file can be any name.
- One parameter file needs to be created for one calculation.
- Parameter name = Value



Sample

▶▶ Reference

For more information about the parameter name, see the descriptions in the later section.

2. Start the command prompt, and enter commands. Alternatively, crate a batch file (*.bat) containing commands, and then execute the commands.

Enter the following commands.

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\inductance_motor\MotorInductanceTool_en.exe
-file="name and path of the parameter file (*.txt)"
-jcf="name and path of the JCF file (*.jcf) of magnetic field
analysis"
```

Example

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\inductance_motor\MotorInductanceTool_en.exe
-file="D:\JMAGData\parameter.txt"
-jcf="D:\JMAGData\FEM\Test.jcf"
```

Important **A command and an argument should be defined in one line. To use two lines, enter "^" (hat) at the end of the line.**

MEMO To use a batch file, enter either of aforementioned command in the text editor, specify a name, and then save. After the extension of the file is changed to (*.bat) and the file is double-clicked, the command is executed.

When the command is executed, the Calculate Motor Inductance tool is executed.

■ Parameters

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
output_csv	String		
current_type	Flag	0 1	Type of current Amplitude/phase d-axis current/q-axis current
current_frequency	Double		Current frequency (Hz)
pole	Int		Number of poles
amplitude_max	Double		Maximum amplitude (A)
amplitude_min	Double		Minimum amplitude (A)
amplitude_division	Int		Divisions for amplitude
phase_max	Double		Maximum value of current phase (deg)
phase_min	Double		Minimum value of current phase (deg)

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
phase_division	Int		Number of divisions for current phase
flg_psim_look_up	Flag	0 1	Do not export the PSIM lookup table Export the PSIM lookup table
flg_remote	Flag	0 1	Execute in the local machine Execute in the remote machines using distribution computing
cpu_group_name	String		Server: Port ID: Group name
flg_result	Flag	0 1	Save the result file of magnetic field analysis Do not save Save
flg_retry	Flag	0 1	Recalculate all Recalculate only failed calculations

12-12-9. Executing Analysis using Third-party Job Management Systems

The calculate motor inductance tool has the following modes built-in.

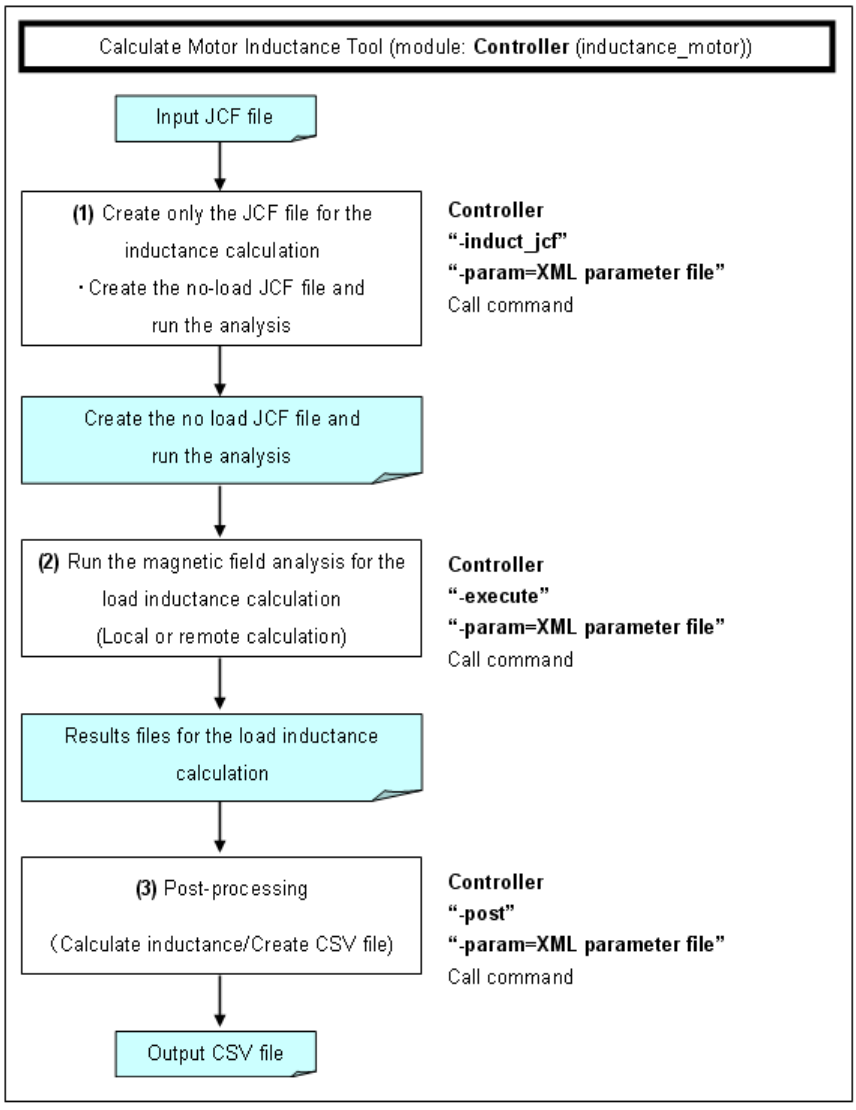
- A mode to bundle and create the input files (*.jcf) used for analyses
- A mode to remotely execute magnetic field analyses using input files (*.jcf)
- A mode for post-processing based on the analysis results

Analysis can be run and managed using a third-party job management system using these modes.

Important **Processes for forwarding input files that are generated and the result files obtained after an analysis as well as adding jobs into a job management system need to be defined by script created by the user.**

■ Call flow

The call flow for job management systems are as follows:


Calculate Motor Inductance Tool (module: Controller (inductance_motor))
Input JCF file
(1) Create only the JCF file for the inductance calculation
・Create the no-load JCF file and run the analysis
Controller
"-induct_jcf"
"-param=XML parameter file"
Call command
Create the no load JCF file and run the analysis
(2) Run the magnetic field analysis for the load inductance calculation (Local or remote calculation)
Controller
"-execute"
"-param=XML parameter file"
Call command
Results files for the load inductance calculation
(3) Post-processing (Calculate inductance/Create CSV file)
Controller
"-post"
"-param=XML parameter file"
Call command
Output CSV file

■ Call command

The following command is executed from the command line to run the calculate motor inductance tool (module: Controller) in a job management system.

```
~/Tools/inductance_motor/Controller.exe
```

In addition, the following argument is required after the command.

```
"-param=[parameter file (*.xml)]"
```

MEMO The "parameter file" creates a folder to save the data when the Calculate Motor Inductance tool is executed. The parameters required to run the Calculate Motor Inductance tool are included in this file in the XML format.

The following argument needs to be added depending on the model.

- Mode to create a JCF file for the Calculate Motor Inductance tool
  Argument: "-induct_jcf"
- Mode to run the magnetic field analysis (local & remote execution)
  Argument: "-execute"
- Mode for post-processing (calculate motor inductance, create CSV file)
  Argument: "-post"

12-13. Calculate Coil Inductance

12-13-1. Function

The coil inductance calculation tool is a convenient tool that can be applied to reactor coils as well as various coil parts.

This tool has the following features.

- Inductance for each current value can easily be obtained by taking into account the nonlinear properties of the material
- Inductance for the stationary current as well as the transient currents that include carrier harmonics.

MEMO The calculate coil inductance tool can be run using a third-party job management system.

12-13-2. Flow of Calculation

The inductance of the coil is calculated in the following steps.

1. **Create an magnetic field analysis model.**

   Set material properties and conditions (FEM coil conditions), and then generate a mesh. A circuit is not necessary.

2. **Start the Calculate Coil Inductance tool, and then import the FEM model created at step 1.**

3. **Specify the parameters for the inductance calculation.**

4. **Run the calculation.**

   Hereinafter, the procedures after step 2 are described.

12-13-3. Files Required to Use the Tool

Before using the tool, prepare the following files.

- JCF file of the magnetic field analysis
  - The JCF files of the magnetic field analyses (transient response analysis and frequency response analysis) are supported.
  - Three or less FEM coil conditions must be set.
  - A circuit is not required.

12-13-4. Notes on the Usage

The following must be noted when using this tool:

- A maximum of three coils can be handled.
- In the calculation of inductance, the magnetic field analyses are run for the number of coils.
- The transient response magnetic field analysis (TR, DP) and frequency response magnetic field analysis (FQ) are supported.

12-13-5. Procedures

1. **Select [Tools] > [Calculate Coil Inductance] from the menu bar.**

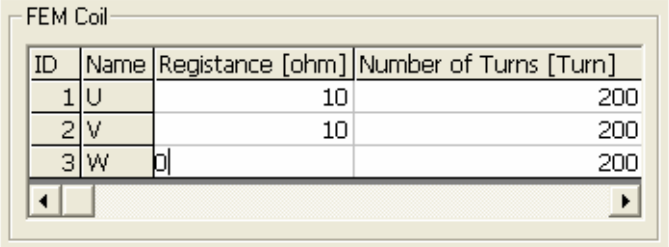
   The [Calculate Coil Inductance] dialog box is displayed.

2. **Click [Browse] of the [JCF File Name] box, and specify the JCF file to reference.**

   When the JCF file is specified, the FEM coil conditions contained in the file are displayed in the table under [FEM Coil].

3. **Click [Browse] of the [Analysis Result Folder] box, and then specify the folder to export the results of the inductance calculation.**

4. **Select each coil in the table, and then enter the resistance value and the number of turns in each cell.**



5. To include the effect of the saturation of magnetic flux, select the [Calculate Inductance with Saturation Flux] check box.

If this option is used, the nonlinearity of the inductance will be included in the calculation.

The equations to calculate the inductance are as follows:

- Nonlinearity not included (check box: OFF)

$$L = \frac{\Phi}{I}$$

- Nonlinearity included (check box: ON)

$$L = \frac{d\Phi}{dI}$$

L: Inductance ϕ: FEM coil flux linkage I: Current

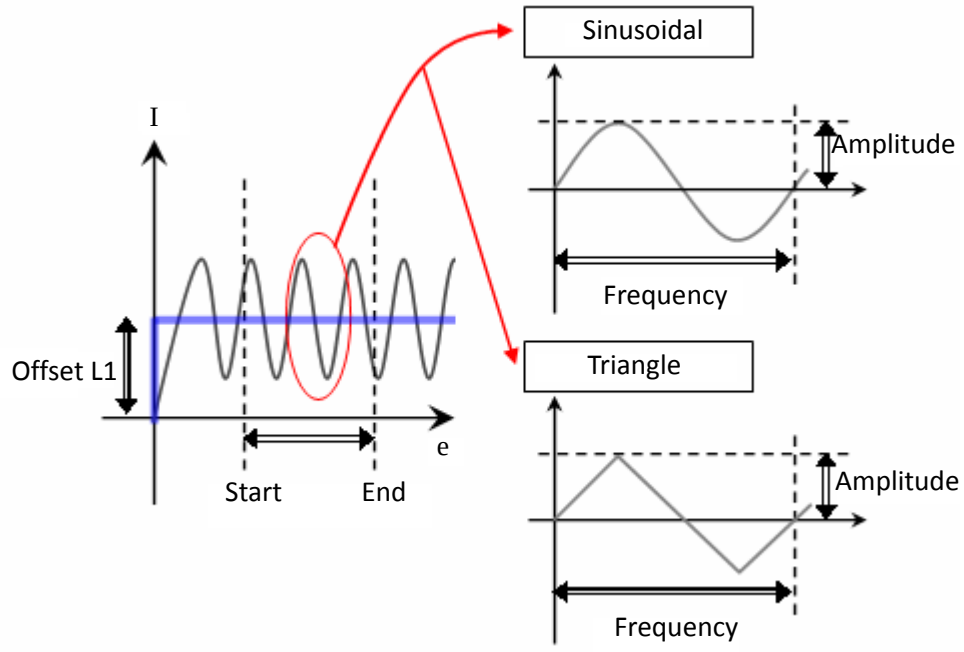
6. Specify the current flowing in the model and the method of inductance calculation.

The available settings vary depending on the type of magnetic field analysis in the reference data. Also, the same settings may have different meaning or effect.

- In the case of the magnetic field analysis (frequency response) (FQ):
  - [Current Value]: Select the specifying method, and then specify the current value.
    - [Constant (A)]: Enter the amplitude. The calculation is run at a zero phase. The inductance is calculated from the current and flux linkage of the FEM coil at zero phase.
    - [Point Sequence]: Enter the point sequence of the amplitude to obtain the inductance at each current value. Click the [Setting] button, and then specify the table in the [Current Point Sequence] dialog box.
- In the case of the magnetic field analysis (transient response analysis) (TR, DP):
  - [Current Value]: Select the specifying method, and then specify the current value.
    - [Constant (A)]: Applies a constant current value.
    - [Point Sequence]: Repeats the transient analysis as many times as the number of data points specified. The inductance is calculated in each analysis for each current value specified in the point sequence. The average time of the FEM coil flux linkage is divided by the current value to obtain the inductance. Click the [Setting] button and enter the current value for each case in the [Current Point Sequence] dialog box.

- [Time]: Specifies the range to be calculated. Set this for calculating a specific range such as when calculating only a steady-state range.
  - [Start Time (sec)]
  - [End Time (sec)]

  If the time is not specified, the average of all steps will be calculated.
- [Carrier Setting]: Specifies the carrier component of the constant current. Setting this allows for the effects of the carrier component on the inductance in an eddy current analysis. The carrier component is superposed on the constant current specified in [Constant (A)] to be applied for the analysis. To set, select the [Include Carrier] check box and then set the following items.
  - [Amplitude (A)]: Enter the current amplitude of the carrier.
  - [Frequency (Hz)]: Enter the frequency of the carrier.
  - [Function Type]: Select the waveform of the carrier.
    - [Sinusoidal]
    - [Triangle]



Function type

MEMO If the motion condition is set for the transient analysis, the Inductance Tool runs the analysis based on the specified conditions, and then calculates the inductance from the FEM coil flux linkage obtained by the analysis and the current value set with the tool. Therefore, if the average flux linkage is zero, the inductance also becomes zero.

7. **To distribute the inductance calculation, select the [Distributed Calculation (Remote)] check box, and then select the CPU group to be used for the calculation from the [CPU Group Name] list.**

   This is the feature that automatically distributes the calculation to the CPU nodes on the JMAG Remote System network.

   For the inductance calculation which has multiple cases due to the variation of current, each case of the calculation is distributed to the CPU nodes.

8. **Select the [Recalculate Failed Cases] check box to recalculate only the cases that failed.**

   The calculation is run up to twice for each failed case.

   Recalculate all cases if the JCF file or the list of cases of the previous calculation cannot be found in the calculation folder.

   MEMO If you select the [Recalculate Failed Cases] option, all other settings cannot be changed.

9. **In case the data created by procedures up to this point is exported to the JCF file, click [Export].**

   In case you wish to conduct inductance calculation on multiple models continuously, after exporting all data to the JCF file, execute the calculation by using the [Execute] dialog box.

10. **Click [Run] to run the inductance calculation.**

    In the [Calculation Status] dialog box, you can monitor the progress of the whole calculation.

    If you select the [Distributed Calculation (Remote)] check box, the [JMAG Remote System Calculation Observer] dialog box appears in which you can monitor the progress of the calculation for each case.

12-13-6. Output Items

- Self inductance (H)
- Mutual inductance (H)

12-13-7. Output Files

- Plot file
- Calculation result file (*.csv) of inductance and coupling coefficient

  *The inductance is defined in (H)

  *The digits represent the IDs of the FEM coils.

$$\begin{pmatrix} L_{11} & M_{12} \\ M_{21} & L_{22} \end{pmatrix} \begin{pmatrix} I_1 \\ I_2 \end{pmatrix} = \begin{pmatrix} \Phi_1 \\ \Phi_2 \end{pmatrix} \qquad \begin{pmatrix} L_{11} & M_{12} \\ M_{21} & L_{22} \end{pmatrix} \begin{pmatrix} dI_1 \\ dI_2 \end{pmatrix} = \begin{pmatrix} d\Phi_1 \\ d\Phi_2 \end{pmatrix}$$

$$K = \frac{\sqrt{|M_{12} \cdot M_{21}|}}{\sqrt{|L_{11} \cdot L_{22}|}}$$

File name: JCF file name.csv

current	L11	L22	L33	M21	M32	M13	M12	M23	M31	K12	K23	K13
Current Value; A	!Self-inductance L11	!Self-inductance L22	!Self-inductance L33	!Matual-inductance M21	!Matual-inductance M32	!Matual-inductance M13	!Matual-inductance M12	!Matual-inductance M23	!Matual-inductance M31	Coupling-coefficient K12	Coupling-coefficient K23	Coupling-coefficient K13

For example, test.csv will be output for test.jcf.

current	L11	L22	L33	M21	M32	M13	M12	M23	M31	K12	K23	K13
1.00E-01	8.25E-02	1.12E-03	8.25E-04	-5.54E-03	-5.56E-04	-2.61E-03	-5.54E-03	-5.56E-04	-2.61E-03	5.77E-01	5.79E-01	3.17E-01
5.00E-01	1.39E-01	1.12E-03	8.25E-04	-1.05E-02	-5.56E-04	-3.28E-03	-5.54E-03	-5.56E-04	-2.61E-03	6.12E-01	5.79E-01	2.74E-01
1.00E+00	2.69E-01	1.12E-03	8.25E-04	-2.19E-02	-5.56E-04	-4.83E-03	-5.54E-03	-5.56E-04	-2.61E-03	6.36E-01	5.79E-01	2.39E-01
3.00E+00	1.95E-01	1.33E-03	9.41E-04	-1.38E-02	-6.61E-04	-5.54E-03	-6.59E-03	-6.52E-04	-2.82E-03	5.94E-01	5.87E-01	2.92E-01
5.00E+00	1.27E-01	2.03E-03	1.39E-03	-8.73E-03	-1.01E-03	-3.85E-03	-1.01E-02	-1.05E-03	-3.28E-03	5.85E-01	6.14E-01	2.69E-01
7.00E+00	9.35E-02	2.98E-03	2.01E-03	-6.34E-03	-1.49E-03	-2.93E-03	-1.49E-02	-1.63E-03	-3.77E-03	5.81E-01	6.35E-01	2.43E-01
1.00E+01	6.72E-02	3.76E-03	2.69E-03	-4.50E-03	-1.88E-03	-2.15E-03	-1.87E-02	-2.20E-03	-4.83E-03	5.78E-01	6.39E-01	2.40E-01

12-13-8. Executing the Tool from the Command Line

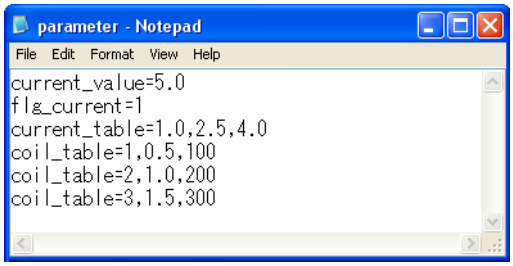
The Calculate Coil Inductance tool can be executed using the command line. The command line is useful for executing multiple files at once.

1. Create a parameter file (*.txt).

Write the change for parameters in the input data created in step 1.

The following should be noted.

- The text format (*.txt) should be used.
- The name of the file can be any name.
- One parameter file needs to be created for one calculation.
- Parameter name = Value



Sample

▶▶ Reference

For more information about the parameter name, see the descriptions in the later sections.

2. Start the command prompt, and then enter commands.

Alternatively, crate a batch file (*.bat) containing commands, and then execute the commands.

Enter the following commands.

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\inductance_coil\Inductance_coil_en.exe
-file="name and path of the parameter file (*.txt)"
-jcf="name and path of the JCF file (*.jcf) of magnetic field
analysis"
```

Example

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\inductance_coil\Inductance_coil_en.exe
-file="D:\JMAGData\parameter.txt"
-jcf="D:\JMAGData\FEM\Test.jcf"
```

Important **A command and an argument should be defined in one line. To use two lines, enter "^" (hat) at the end of the line.**

MEMO To use a batch file, write either of aforementioned command in the text editor, specify a name, and then save. Change the extension of the file to (*.bat), and then double-click the file, the command is executed.

When the command is executed, the calculate coil inductance tool is executed.

■ Parameters

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
output_directory	String		Directory of the analysis result file
coil_table	Table		FEM coil ID, resistance, number of coil turns
flg_current	Flag	0 1	Type of current Constant Point sequence
current_value	Double		Current value (constant)
current_table	Table		Point sequence for current Double needs to be separated by commas.
flg_carrier	Flag	0 1	Specify the carrier Do not allow Allow (available for transient response analysis)

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
carrier_amplitude	Double		Current amplitude when the carrier is taken into account (A)
carrier_frequency	Double		Current frequency when the carrier is taken into account (Hz)
flg_carrier_type	Flag	0 1	Function type Sinusoidal Triangular wave
flg_saturation	Flag	0 1	Do not calculate inductance for magnetic saturation Calculate inductance for magnetic saturation
flg_time	Flag	0 1	Time Specify Do not specify
start_time	Double		Start time (sec)
end_time	Double		End time (sec)
flg_remote	Flag	0 1	Execute in local machine Execute in the remote machines using distribution computing
cpu_group_name	String		Server: Port ID: Group name
flg_retry	Flag	0 1	Recalculate all Recalculate only failed calculations

12-13-9. Executing Analysis using Third-party Job Management Systems

The calculate coil inductance tool has the following modes built-in.

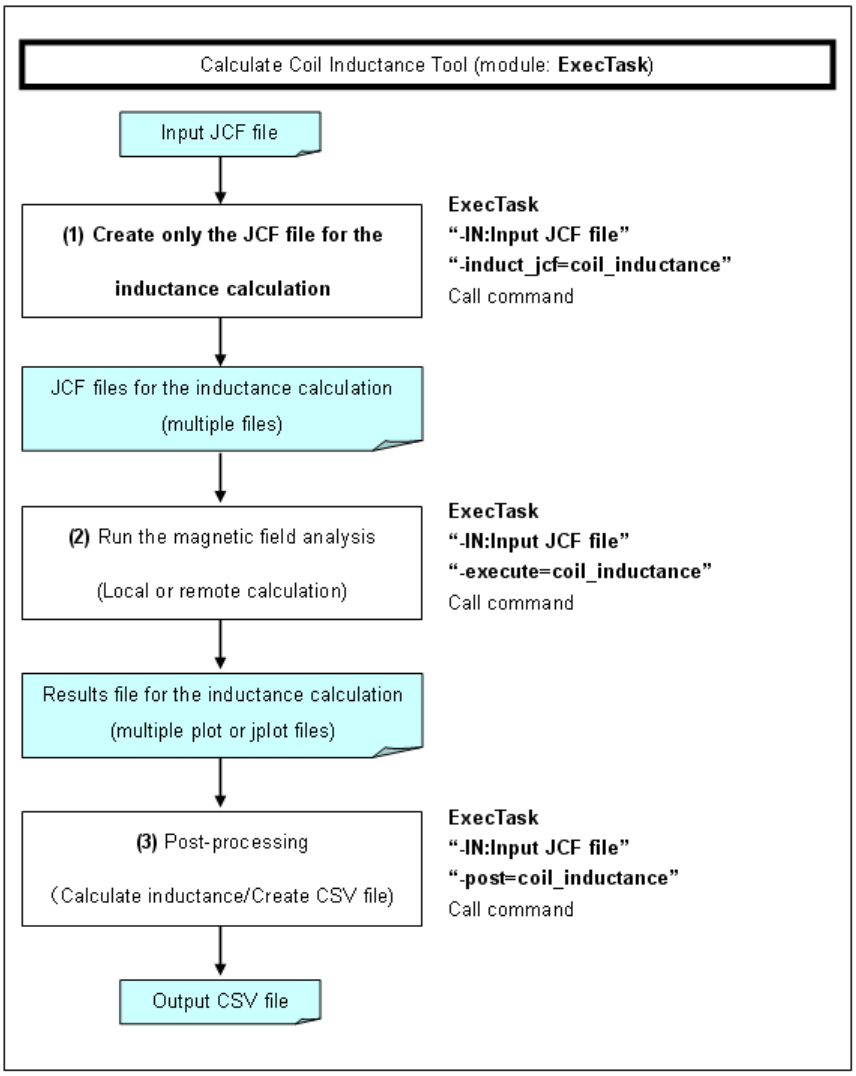
- A mode to bundle and create the input files (*.jcf) used for analyses.
- A mode to remotely execute magnetic field analyses using input files (*.jcf).
- A mode for post-processing based on the analysis results.

Analysis can be run and managed using a third-party job management system using these modes.

Important **Processes for forwarding input files that are generated and the result files obtained after an analysis as well as adding jobs into a job management system need to be defined by script created by the user.**

■ Call flow

The call flow for third-party job management systems are as follows:



■ Call command

The following command is executed from the command line to run the calculate coil inductance tool (module: ExecTask) in a job management system.

```
~/ExecTask.exe
```

The following argument needs to be added depending on the model.

- Mode to create a JCF file for the calculate coil inductance tool
  Argument: “-induct_jcf=coil_inductance”
- Mode to run the magnetic field analysis (local & remote execution)
  Argument: “-execute=coil_inductance”
- Mode for post-processing (calculate coil inductance, create CSV file)
  Argument: “-post=coil_inductance”

12-14. Calculate Busbar Inductance

12-14-1. Function

In order to calculate the Inductance of the Elec Conductor, the inside of the conductor is divided into several parts, and the inductances of each part as well as between the parts are calculated. This is an effective tool for shape designing of bus bar, etc.

In order to use this function, besides 3D magnetic field analysis solver (FQ), an exclusive license will be necessary.

MEMO The calculate busbar inductance tool can be run using a third-party job management system.

12-14-2. Flow of Calculation

The calculation of partial inductance is executed in accordance with the following flow.

1. **Start up the project of 3D frequency response analysis**
2. **Conduct operations up to material setting (electrical properties), condition setting (FEM conductor condition, partial inductance condition), and mesh creation.**
3. **Export the created data to JCF file.**
4. **Start up the busbar inductance tool.**
5. **Specify the JCF file that was created by Procedure 3) as Input file.**
6. **Specify parameters such as frequency.**
7. **Execute inductance calculation.**

An explanation will be given below on Procedure 4) and after.

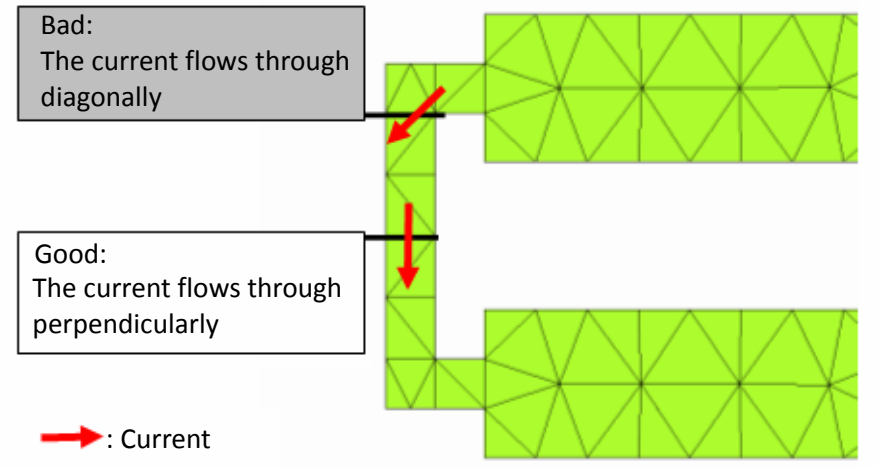
12-14-3. Files Required to Use the Tool

■ JCF file of 3D frequency response magnetic field analysis

Prior to using the busbar inductance tool, create a JCF file with 3D frequency response analysis project. At this time, pay attention to the following points.

- Secure sufficient air layer around the Elec Conductor.
- Calculation regions are reckoned in material units. Even if the material object is separated, if the same material (for instance, material named No. 1), from the standpoint of the tool is set, it will be reckoned as one calculation region. Even if they are parts having the same physical properties, in case there are places that will be calculated and places that will not be calculated, set different materials.
- The model that has a magnetic material cannot be calculated with this tool.
- Settings of frequency conditions in the JCF file will be neglected. The frequency set in the busbar inductance tool will be used in the analysis.
- It is necessary that the partial inductance calculation conditions are set.
- It is necessary that the FEM conductor conditions are set.
- There is no need of creating a circuit.
- The JCF file for mesh data should be used. The JCF file for geometry cannot be used.
- The inductance calculation tool cannot be used with the following functions.
  - Adaptive mesh function
  - Patch mesh function (function to generate mesh at each step and motion condition)
  - Layered mesh
  - Eccentricity function for motion (rotation) condition

- In case the electric current does not pass through the inflow port plane of each part perpendicularly, there is the possibility of the current and the Inductance not being calculated correctly. At the time of creating models and specifying ports, put this matter into consideration. (See the following drawing.)



Setting of flow-in port

12-14-4. Procedures for Executing the Tool

1. **From the menu bar, select [Tools] > [Calculate Busbar Inductance].**

   The [Calculate Busbar Inductance] dialog box will appear.

2. **Click [Browse] of the [Input File] text box, and specify the JCF file of 3D frequency response magnetic field analysis.**

   A list of calculation regions to which partial inductance conditions are set to the [Setting of Current Path] list.

3. Select the calculation region from the [Setting of Current Path] list, and enter the amplitude and phase for each cell.

Against all parts displayed in the list, specify the amplitude and phase.

Furthermore, an equation that shows the relation among current value, amplitude and phase is as follows:

$$\mathrm{Current[A]} = \mathrm{I} \times \sin\left(\theta + \frac{\mathrm{P}\pi}{180}\right)$$

I:Amplitude (A)

P:Phase (deg)

4. Specify the frequency range in which Inductance calculation is conducted.

- [First Frequency (Hz)]
- [Final Frequency (Hz)]
- [Number of Divisions]: Specify the number of divisions to be made to the range defined by the start frequency and the end frequency.
- [Division Method]: Select from the following two types.
  - [Regular Interval]
  - [Logarithmic Division]

5. Click [Setting of Inductance Matrix], and open the [Induction matrix] dialog box. Subsequently, select only the check boxes of the combination of calculation regions for which calculation will be executed.

6. In case inductance calculation is not performed when the distance between calculation regions exceeds the standard value, select the [Upper Limit Distance of Parts (m)] check box and enter the distance to be used as judgment standard in the text box.

The distance of the parts mentioned here means the distance between the center of gravity of a certain part and the center of gravity of the other part.

In case this distance exceeds the standard value, calculation of that combination of Calculation Regions will not be executed.

Settings of the [Inductance matrix] dialog box of Procedure 4) supersede the specification of the [Upper Limit Distance of Parts (m)] check box.

7. In case inductance calculation is not performed when the amount of current flowing through the calculation region is below the standard value, select the [Lower Limit of the Current Value (%)] check box and enter the value to be used as judgment standard in the text box.

If the value calculated by the following formula is below the standard value, the self-inductance of that calculation region cannot be calculated.

$$\text{Current of each calculation region [\%]} = \frac{\text{Current of each calculation region [A]}}{\text{Maximum value of current flowing through the model [A]}}$$

Settings of the [Inductance matrix] dialog box of Procedure 4) supersede the specification of the [Lower Limit of the Current Value (%)] check box.

8. To distribute the inductance calculation, select the Distributed Calculation (Remote Execute) check box, and then select the CPU group to be used for the calculation from the [CPU Group Name] list.

This is the feature that automatically distributes the calculation to the CPU nodes on the JMAG Remote System network.

In an inductance calculation, multiple cases due to the variation in current are calculated. The calculation is distributed by the case.

9. Select the [Only calculation which went wrong is re-performed] check box to recalculate only the cases that failed.

The calculation is run up to twice for each failed case.

Recalculate all cases if the JCF file or the list of cases of the previous calculation cannot be found in the calculation folder.

MEMO If you select the Recalculate Failed Cases option, all other settings cannot be changed.

10. In case the data created by the procedures up to this point are exported to JCF file, click the [Write] button.

In case Inductance calculations are executed on multiple models continuously, execute calculation by using the [Execute] dialog box after exporting all data to the JCF file.

11. Click the [Execute] button.

In the [Calculation Status] dialog box, you can monitor the progress of the whole calculation.

If you select the [Distributed Calculation (Remote Execute)] check box, the [JMAG Remote System Calculation Observer] dialog box appears in which you can monitor the progress of the calculation for each case.

12-14-5. Output Items

- Resistance (Ohm)
- Self inductance (H)
- Mutual inductance (H)

12-14-6. Output Files

- PLOT file
  Calculation result file (*.csv) of inductance and resistance
  File name: busba.csv

Datatype	Solver type of magnetic field analysis	
Filename	Name of result file	
Frequency[Hz]	Frequency value	
	1:conductor1	2:conductor2
Current[A]	Current value of conductor1	Current value of conductor2
Registance[ohm]	Resistant value of conductor1	Resistant value of conductor2
Inductance[H]	1:conductor1	2:conductor2
1:conductor1	Self-inductance of conductor1	Mutual inductance between conductor1and
2:conductor2	Mutual inductance between conductor1and	Self-inductance of conductor2

Sample:

Datatype	JMAG-Studio. FQ.	
Filename	busbar.plot	
Frequency[Hz]	1.00E+03	
	1:conductor1	2:conductor2
Current[A]	1	1
Registance[ohm]	3.21E-04	3.00E-04
Inductance[H]	1:conductor1	2:conductor2
1:conductor1	3.66E-08	-3.03E-08
2:conductor2	-3.03E-08	3.70E-08

12-14-7. Executing from the Command Line

The Calculate Busbar Inductance tool can be executed using the command line.

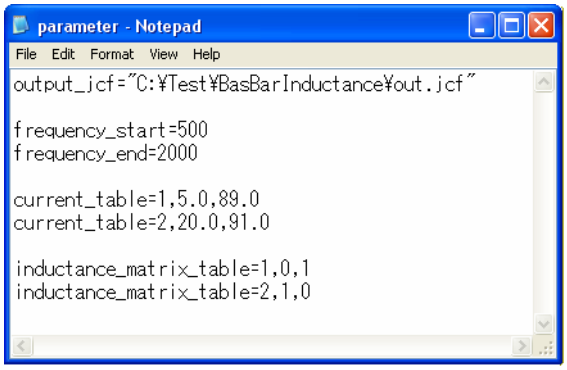
The command line is useful for executing multiple files at once.

1. Create a parameter file (*.txt).

Write the change for parameters in the input data created in step 1.

The following should be noted.

- The text format (*.txt) should be used.
- The name of the file can be any name.
- One parameter file needs to be created for one calculation.
- The parameter name and its value need to be specified as “Parameter name = Value.”



Sample

▶▶ Reference

For more information about the parameter name, see the descriptions in the later sections.

2. Start the command prompt, and enter commands.

Alternatively, crate a batch file (*.bat) containing commands, and then execute the commands.

Enter the following commands.

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\inductance_busbar\Inductance_busbar.exe
-file="name and path of the parameter file (*.txt)"
-jcf="name and path of the JCF file (*.jcf) of magnetic field
analysis"
```

Example

```
cd "C:\Program Files\JMAG-Designer12.0"
Tools\inductance_busbar\Inductance_busbar.exe
-file="D:\JMAGData\parameter.txt"
-jcf="D:\JMAGData\FEM\Test.jcf"
```

Important **A command and an argument should be defined in one line. To use two lines, enter "^" (hat) at the end of the line.**

MEMO To use a batch file, enter either of aforementioned command in the text editor, specify a name, and then save. After the extension of the file is changed to (*.bat) and the file is double-clicked, the command is executed.

When the command is executed, the Calculate Busbar Inductance tool is executed.

■ Parameters

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
output_jcf	String		Name of the JCF file to export
current_table	Table (Int, Double, Double)		Current path (ID/Name/Amplitude/phase)
frequency_start	Double		First frequency (Hz)
frequency_end	Double		Final frequency (Hz)
frequency_division	Int		Number of divisions
flg_division_type	Flag	0 1	Division method Regular interval Logarithm division

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
inductance_matrix_table	Table (Int, Int, Int)		Setting of inductance matrix (ID of combination of calculation regions, ID of calculation region, flag for calculation target (0: Do not calculate, 1: Calculate))
flg_part_distance	Flag	0 1	Upper limit of distance of parts Use Do not use
part_distance_max	Double		Value for the upper limit of distance of parts (m) Use if flg_part_distance=1
flg_current	Flag	0 1	Lower limit of current value Do not use Use
current_min	Double		Lower limit of current value (%) Use if flg_current=1
flg_remote	Flag	0 1	Execute in the local machine Execute in the remote machines using distribution computing
cpu_group_name	String		Server: Port ID: Group name For example, 10.157.102.8888:test1
flg_retry	Flag	0 1	Recalculate all Recalculate only failed calculations

12-14-8. Executing Analysis using Third-party Job Management Systems

The calculate busbar inductance tool has the following modes built-in.

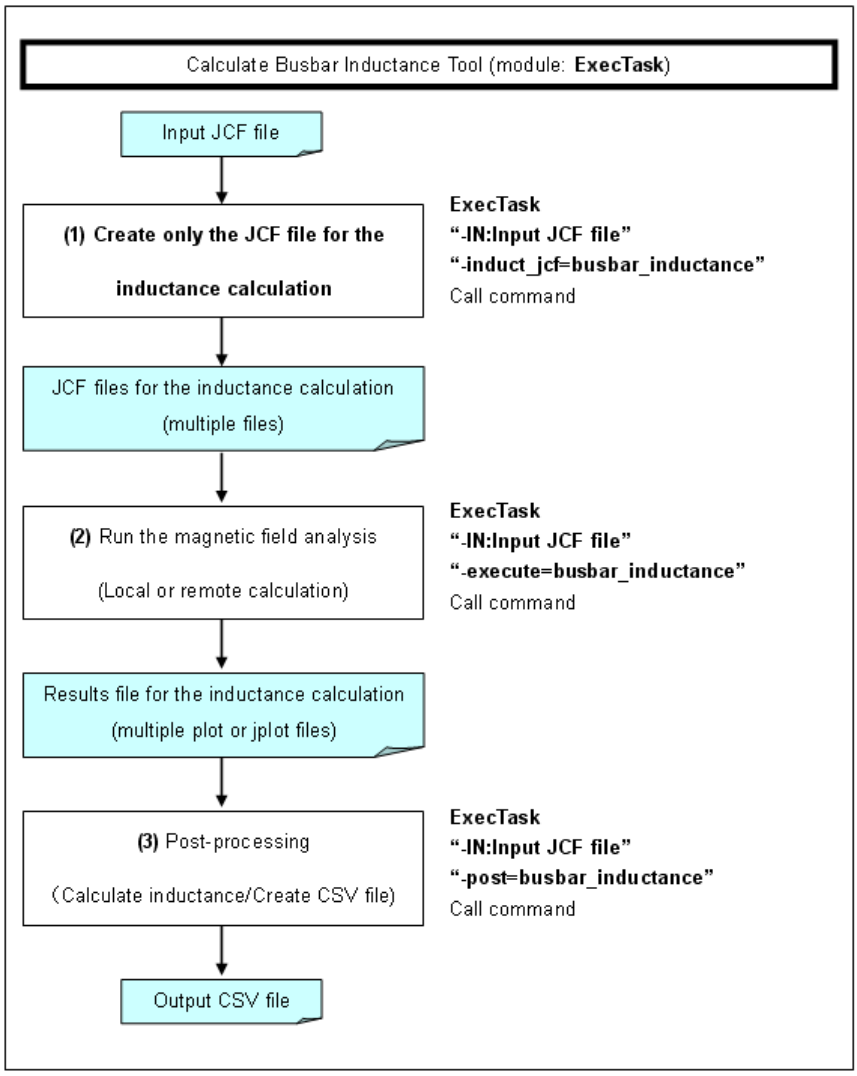
- A mode to bundle and create the input files (*.jcf) used for analyses.
- A mode to remotely execute magnetic field analyses using input files (*.jcf).
- A mode for post-processing based on the analysis results.

Analysis can be run and managed using a third-party job management system using these modes.

Important **Processes for forwarding input files that are generated and the result files obtained after an analysis as well as adding jobs into a job management system need to be defined by script created by the user.**

■ Call flow

The call flow for third-party job management systems are as follows:



Call command

The following command is executed from the command line to run the calculate busbar inductance tool (module: ExecTask) in a job management system.

```
~/ExecTask.exe
```

The following argument needs to be added depending on the model.

- Mode to create a JCF file for the calculate busbar inductance tool
  Argument: “-induct_jcf=busbar_inductance”
- Mode to run the magnetic field analysis (local & remote execution)
  Argument: “-execute=busbar_inductance”
- Mode for post-processing (calculate busbar inductance, create CSV file)
  Argument: “-post=busbar_inductance”

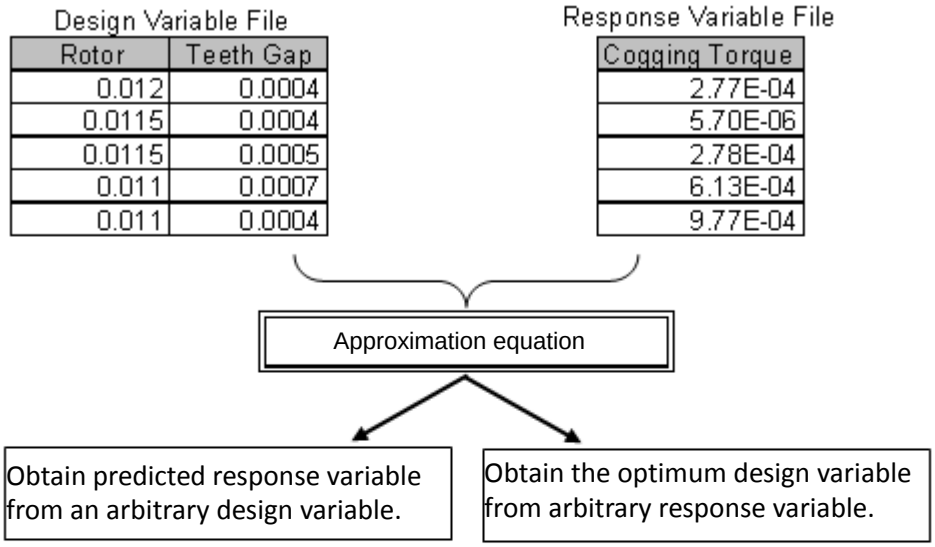
12-15. Calculate Design Approximate (JMAG-Studio)

12-15-1. Function

This function prepares an approximation equation from the combination of design variables and response variables obtained by the analysis, and based on this equation, calculates the response variables predicted from the arbitrary design variables.

In addition, it calculates an optimum design variable for obtaining an arbitrary response variable based on the same approximation equation.

By utilizing this function, it is possible to evaluate the results approximately without executing analysis based on the finite element method over and over.



Example 1)
In the case of the design variable file and response variable file shown in the above drawing, if "ROTOR: 1.15e-2" and "TEETHGAP: 5.5e-4" are specified with design variables, "Cogging Torque: 2.5051250096e-4" is calculated as the predicted response variable.

Example 2)
In the case of the design variable file and response variable file shown in the above drawing, if "Cogging Torque: 2.5e-4" is specified with response variable "ROTOR: 1.1985e-2 (sensitivity: 1.7106440032)" and "TEETHGAP 4e-4 (sensitivity: 4.4164899924)" is calculated as the optimum design variable.

Conceptual drawing of design approximate calculation tool

12-15-2. Files Required to Use the Tool

To use this function, the following two files will be needed.

- Design variable file (CSV file (*.csv))

  In case the variables are set using the parametric tools, click [Variable Output] of the [JMAG-Parametric Tool] dialog box.

  ▶▶ Reference

  For more information about the parametric analysis, refer to JMAG-Studio User's Manual, “Chapter 14, Parametric Analysis”.

  Furthermore, it is also possible to use an arbitrarily prepared CSV file as a design variable file. In such a case, pay attention to the following points.

  - From the standpoint of calculation accuracy, in case n pieces of design variables exist, we recommend you prepare data of 2n + 1 piece or more.

A	B	C
a1	b1	c1
a2	b2	c2
a3	b3	c3
a4	b4	c4
a5	b5	c5
a6	b6	c6
a7	b7	c7

Example: Design variable file

The table on the left shows the three types of variables (A, B, C). As a result, (2 x 3) + 1 = 7.

In other words, for each variable, if there are seven rows of data, a high accuracy calculation can be obtained.

- Response variable file (CSV file (*.csv))

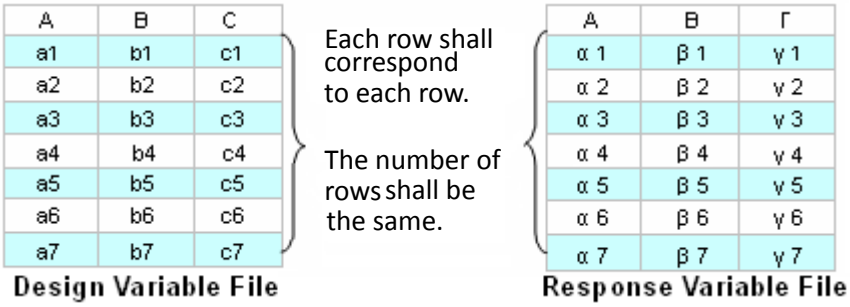
  This file can be prepared by clicking [Output Property] of the [Graph Control] dialog box of the analysis result picture.

  ▶▶ Reference

  Please see the JMAG-Studio User's Manual, “Chapter 7, Displaying Analysis Results” “Option Menu”.

  In addition, an arbitrarily prepared CSV file can also be used as a response variable file. In such a case, pay attention to the following points.

- In the response variable file, data of the same number of rows as the design variable file is necessary. For instance, when data equivalent to seven rows is contained in the design variable file, the response variable file also requires data equivalent to seven rows.



- The design variable file and response variable file must correspond to each other row for row.

12-15-3. Procedures for Executing the Tool

1. **From the menu bar in JMAG-Studio, select [Tools] > [Calculate Design Approximate].**

   The [Calculate Design Approximate] dialog box will appear.

2. **Click [Load].**

   The [Load] dialog box will appear.

3. **Specify the load design variable file and response variable file, and click [OK].**

   Click the ... button next to each box, open the [Open] dialog box, and specify the file. Select a file that is in the specified format, click [OK], and the dialog box returns to the [Design Approximate Calculation Tool] dialog box. Then the approximation equation will be calculated automatically. If an error message is indicated, specify the design variable file and the response variable file again in accordance with the contents of the message.

4. Obtain the approximate value of the response variable or the optimum value of the design variable.

- To obtain the response variable predicted from a design variable: Select the [Approximation] radio button in the [Calculation Mode]. Consecutively, click each cell of the [Value] column in the [Design Variable] list, and input a value.
  When the [→] button is clicked, the response variable predicted from the design variable will be displayed in each cell of the [Value] column in the [Response Variable] list.
  Furthermore, values will also be displayed in the [Sensitivity] column in the [Design Variable] list. As to which design variable the sensitivity corresponds to, that will be displayed in the [Response Variable to Observe] combo box. If you want to obtain the sensitivity of other design variables, make selections from the [Response Variable to Observe] combo box.
- To obtain optimum design variable from an arbitrary response variable: Select the [Optimization] radio button in the [Calculation Mode] dialog box. Consecutively, select the type of response variable specified from the [Response Variable to Observe] combo box, and input a value to the [Desired Value] text box.
  When the [←] button is clicked, the optimum design variable for obtaining the specified response variable is displayed in each cell of the [Value] column in the [Design Variable] list.
  In addition, at the same time, values will also be displayed in the [Sensitivity] column of the [Design Variable] list.

MEMO "Sensitivity" indicates "the degree of influence that changes in a certain design variable has on the response variable." "High sensitivity" means "when a small change is made to the design variable, the response variable will change significantly." On the contrary, "low sensitivity" means "when a small change is made to the design variable, hardly any changes occur to the response variable."

5. To export the design variables and the response variables displayed in the dialog box to a file, click [Export], open the [Save As] dialog box, and save them under a name.

The file is exported is in the CSV format (*.csv).

6. When the work is completed, click [Exit].

The [Calculate Design Approximate] dialog box will close.

Chapter 13. JMAG-RT

This chapter describes the JMAG-RT tool that is provided with JMAG.

13-1. About JMAG-RT

JMAG-RT is a tool to create analysis data based on the results of a JMAG Finite Element Method (FEM) analysis to use in a circuit/control simulator.

Analyzing the various characteristics of a motor (geometry, materials, etc.) is simple if a circuit/control simulation is performed using a motor model that includes an equivalent circuit.

Furthermore, an analysis that accounts for the motor characteristics can also be performed by directly linking JMAG and the circuit/control simulator, however, the time required for an analysis increases largely because the FEM analysis is performed for each step.

JMAG-RT can run high-speed circuit/control simulations while taking into account the motor characteristics obtained from an FEM analysis.

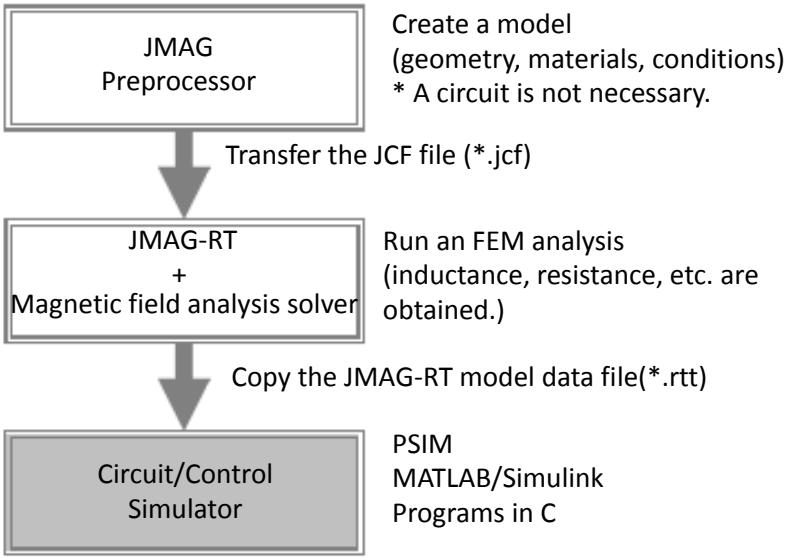
JMAG-RT creates specialized data (hereinafter referred to as a JMAG-RT model) for the circuit/control simulator by obtaining the required motor parameters (linkage flux of coils, inductance, resistance, etc.) from the FEM analysis results of the motor. The motor characteristics can be taken into account in a circuit/control simulation by referring to a JMAG-RT model.

The JMAG RT model can be used as a circuit component in a circuit/control simulation. Therefore, a JMAG FEM analysis that includes a circuit and a circuit/control simulation do not need to be performed each time the drive circuit is modified.

A parametric FEM analysis is performed in JMAG to obtain the circuit characteristics of the model in JMAG-RT. The parametric analysis requires some time, but the amount of time that is required can be reduced by distributing the calculations using the JMAG-Remote System.

A circuit/control simulation can be performed using JMAG-RT as follows.

1. **Create an FEM model with the JMAG preprocessor.**
2. **Run an FEM analysis with the magnetic field analysis solver, and then create a JMAG-RT model in JMAG-RT.**
3. **Run a circuit/control simulation using the JMAG-RT model.**



MEMO A JMAG-RT model is exported as a JMAG-RT model data file (*.rtt). The basic information contained in the file can be checked in the JMAG-RT Viewer.

▶▶ Reference

For more information about the JMAG-RT Viewer, see "Chapter 14. JMAG-RT Viewer" on page 1187.

MEMO The parameters to create a JMAG-RT model and copy the JMAG-RT model to a circuit/control simulator are specified using the JMAG-RT Library Manager in JMAG-RT.

MEMO A feature equivalent to the Calculate Motor Inductance tool in JMAG-Studio Version 9.1 has been added to JMAG-RT. Select a three-phase PM synchronous motor (Ld, Lq model) to use this calculation feature.

MEMO A third-party job management system can be used for calculation processes of JMAG-RT.

▶▶ Reference

For more information about the JMAG-Remote System, see the JMAG-Remote System Installation Manual.

13-2. Models Available in JMAG-RT

JMAG-RT supports the following models.

- Three-phase PM synchronous motor
- Two-phase stepping motor (monopolar)
- Linear solenoid
- Linear synchronous motor
- SR motor
  * Coupling with Simulink and C programs are supported. Coupling with PSIM is not supported
- Induction motor
  * Coupling with Simulink and C programs are supported. Coupling with PSIM is not supported

Reference

See following chapters for details.

- P.1036 "13-8. Three-phase PM Synchronous Motor"
- P.1073 "13-9. Two-phase Stepping Motor"
- P.1090 "13-10. Linear Solenoid"
- P.1104 "13-11. Linear Synchronous Motor"
- P.1126 "13-12. SR Motor"
- P.1139 "13-13. Induction Motor"

13-3. Programs Required for JMAG-RT

The following programs needs to be installed to use this function.

13-3-1. Create a JMAG-RT Model (running the FEM Analysis with JMAG-RT)

- JMAG-RT
  - A JMAG-RT license needs to be installed to run JMAG-RT.
- Magnetic Field Analysis Solver
  - A DP license needs to be installed when using a 2D FEM model.
  - A TR license needs to be installed when using a 3D FEM model.
  - A FQ license is required regardless of the model dimension when calculating an induction motor.

13-3-2. Run the Circuit/Control Simulation using a JMAG-RT Model

■ Use MATLAB/Simulink

- MATLAB/Simulink
  - Use versions R2006b or later for 32bit version.
  - Use versions R2007b or later for 64 bit version.
- SimPowerSystems

MEMO SimPowerSystems is not required for a simulation using the voltage signal input or current signal input of Simulink.

■ Use PSIM

- PSIM (7.1.2 or a later version) and MagCoupler-RT option
  - PSIM 8.0.7 and earlier versions are supported by JMAG-Designer 10.1.1 or earlier versions, or JMAG-Studio 10.01301 or earlier versions.
  - PSIM 9.0.1 and later versions are supported by JMAG-Designer 10.1.2 or later versions, or JMAG-Studio 10.01401 or later versions.

The files that are compatible between PSIM 9.0.1 and JMAG are as follows:

- The JMAG-RT block contained in the file (*.sch) created with PSIM 8.0.7 or earlier versions can be used. In this case, reference the XML file exported from Designer 10.1.1 or earlier versions, or JMAG-Studio 10.01301 or earlier versions.
- The JMAG-RT block contained in a new file (*.psimsch) can be used. In this case, reference the XML file exported from Designer 10.1.2 or later versions, or JMAG-Studio 10.01401 or later versions.
- The JMAG-RT block contained in an older file (*.sch) cannot be used for a new file (*.psimsch).

Important **This function is not supported by the SR motor and Induction motor.**

MEMO PSIM 8.0.4 or a later version is required to use a spatial harmonic model for a three-phase PM synchronous motor.

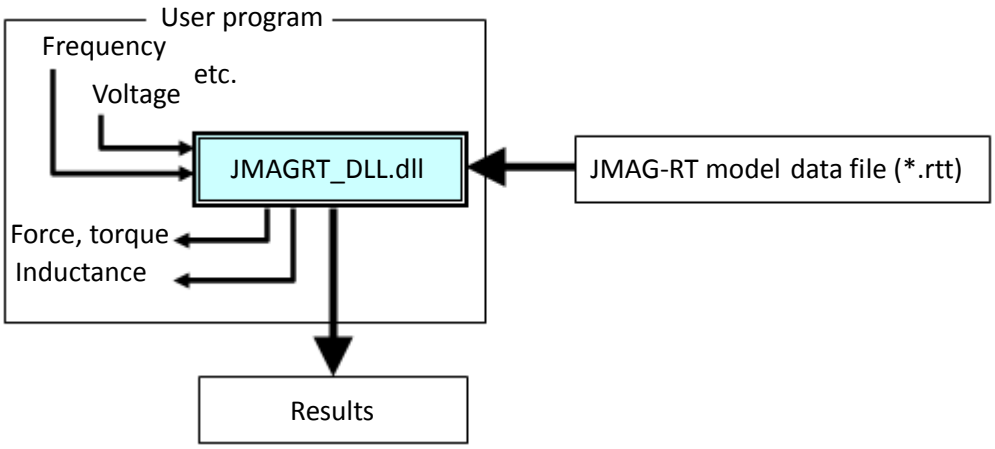
MEMO About PSIM MagCoupler-RT

The MagCoupler-RT option is supported by PSIM 6.2 or a later version. A block for a JMAG-RT model is added to the PSIM library with this option.

Version	Motor Type	Block
-PSIM 9.0.0	Three-phase PM synchronous motor, linear synchronous motor	MagCoupler-RT PMSM Machine block
	Stepping motor, linear solenoid	MagCoupler-RT block
PSIM 9.0.1-	Three-phase PM synchronous motor	PM Synchronous Machine (RT) block
	Linear synchronous motor	Linear Synchronous Machine (RT) block
	Stepping motor	Step Machine (RT) block
	Linear solenoid	Linear Solenoid (RT) block

■ Use Programs in C

- A program in C
  The programs in C need to be created based on the program structure described later.
  A program in C can also be created using a sample file that is provided with JMAG.



Simulation using programs in C

MEMO JMAGRT_DLL.dll is a a dynamic link library (DLL) to control a simulation using a program in C.

▶▶ Reference

- For information about the JMAGRT_DLL.dll file, see P.1170 "13-15-1. Flow of Simulations Using Programs in C".
- For more information about simulations using programs in C, see P.1170 "13-15. Running Simulations Using Programs in C".

13-4. Using JMAG-RT (JMAG-RT Library Manager)

1. Create an FEM model with JMAG

Create an FEM model with JMAG-Designer or JMAG-Studio, and then export the FEM model data to a JCF file.

The FEM model used to create a JMAG-RT model in JMAG-RT must satisfy several requirements. If these requirements are not satisfied, the FEM model cannot be imported into the JMAG-RT Library Manager. The requirements differ according to the type of model.

MEMO For three-phase PM synchronous motors, it is possible to create a JMAG-RT model from data written in CSV format. In this case, it is not necessary to create FEM model.

▶▶ Reference

- See also "13-6. Creating JMAG-RT Model from a CSV File" on page 1033.
- See following chapters about notes about creating FEM model.
  - "13-8. Three-phase PM Synchronous Motor" on page 1036
  - "13-9. Two-phase Stepping Motor" on page 1073
  - "13-10. Linear Solenoid" on page 1090
  - "13-11. Linear Synchronous Motor" on page 1104
  - "13-12. SR Motor" on page 1126
  - "13-13. Induction Motor" on page 1139

2. Start the JMAG-RT Library Manager

The parameters to create a JMAG-RT model and copy the JMAG-RT model to a circuit/control simulator is specified in the JMAG-RT Library Manager.

The JMAG-RT Library Manager can be started as follows.

■ JMAG-Studio

i. Select [Start] > [All Programs] > [JMAG-Studio] > [JMAG-RT] from the Windows start menu.

■ JMAG-Designer

i. Select [Tools] > [JMAG-RT] from the menu bar.

The JMAG-RT Library Manager startup window is displayed.

3. Import the FEM model data into the JMAG-RT Library Manager .

Click [New] in the JMAG-RT Library Manager startup window, and then select the type of the model from the [Model Type] box. The following types of models can be selected.

- [Three-phase PM Synchronous Motor]
- [Two-phase Stepping Motor]
- [Linear Solenoid]
- [Linear Synchronous Motor]
- [SR Motor]
- [Induction Motor]

■ In the case of selecting the three phase PM synchronous motor in [Model Type] box

Click [OK] button then [Select Motor Model] dialog is shown. The following two options can be selected.

- [Execute FEM Calculation]: This option should be selected to create a JMAG-RT model from the result of FEM analysis by JMAG. This option requires following settings.
  - [JCF File]: Click [Browse] button to select JCF file for FEM analysis in JMAG-RT.
  - [Ld, Lq Model], [Spatial Harmonic Model]: Turn on the check box suitable for your required accuracy and purpose.

  Important **Turn on the [Ld, Lq Model] check box if it will be required to display the draw torque-speed curve or efficiency map on JMAG-RT.**

  ▶▶ Reference
  For more information about the JMAG-RT Viewer, see "Chapter 14. JMAG-RT Viewer" on page 1187.
- [User data]: This option should be selected to create JMAG-RT model from user prepared data written in CSV format. This option requires following setting.
  - [CSV File]: Click [Browse] button to select CSV file.

  ▶▶ Reference
  See "13-6. Creating JMAG-RT Model from a CSV File" on page 1033.

Click [OK] after specifying the settings. The setup dialog box is displayed.

■ In the case of selecting other motor types in [Model Type] box except three phase PM synchronous motor

Click [Browse] in the [Model Type] dialog box, and then specify the JCF file exported in Step 1. Click [OK]. The setup dialog box specific to the type of model is displayed.

4. Specify the settings for the FEM model with the JMAG-RT Library Manager to create a JMAG-RT model.

The settings vary according to the type of model.

▶▶ Reference

See following chapters for details.

- "13-8. Three-phase PM Synchronous Motor" on page 1036
- "13-9. Two-phase Stepping Motor" on page 1073
- "13-10. Linear Solenoid" on page 1090
- "13-11. Linear Synchronous Motor" on page 1104
- "13-12. SR Motor" on page 1126
- "13-13. Induction Motor" on page 1139

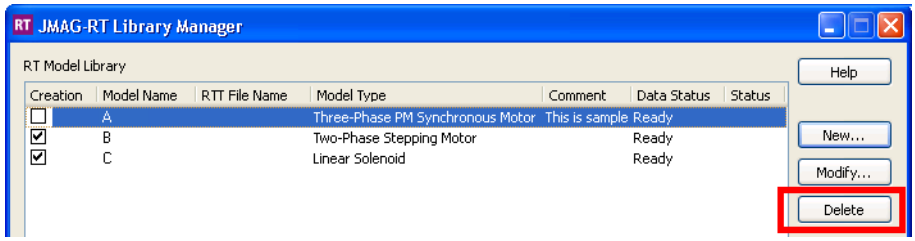
Click [OK] to return to the startup window when the setup is complete.

The FEM model (hereinafter called a base model) is imported to the [RT Model Library] list.

Select the base model from the list, and then click [Modify] to change the settings specified for a base model in the list.

Select the base model from the list, and then click the [Delete] button to delete a base model.

MEMO A base model is not modified or deleted if the [Creation] check box is selected in the [RT Model Library] list. For example, only A is deleted when [Delete] is clicked if the [RT Model Library] list is the same as the figure below.



Use caution when modifying or deleting base models. The [Creation] check box is used for the following.

- Select a base model to create a JMAG-RT model.
- Select a JMAG-RT model to copy to the circuit/control simulator.

MEMO A base model is locked so that it cannot be edited after a JMAG-RT model is created in the JMAG-RT Library Manager. All of the base models that have "Created" displayed in the [Data Status] column in the [JMAG-RT Model Library] are locked. A message asking whether to unlock the protected data is displayed when [Modify] is clicked and the parameters are changed for a model that is locked. Clicking [Yes] unlock the protected data.
A base model that has "Ready" displayed in the [Data Status] column has all of the necessary settings specified, but a JMAG-RT model has not been created, or an JMAG-RT model has been created, but the data has been unlocked.

MEMO The status of a JMAG-RT model that is being created is displayed in the [Status] column.

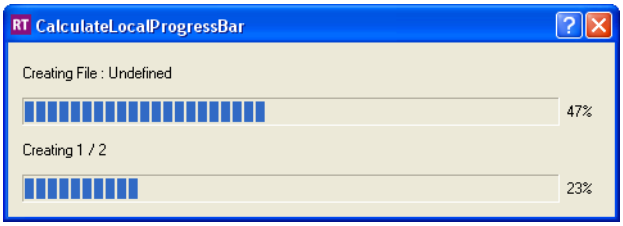
5. Create a JMAG-RT model (running the FEM analysis with JMAG-RT).

Select the [Creation] check box for a base model to create a JMAG-RT model, and then click [Create Model]. The FEM analysis starts running for multiple cases to create a JMAG-RT model.

A progress bar is displayed in the [Calculate Local Progress Bar] dialog box during the analysis.

MEMO The analyses are run in order from the top of the [RT Model Library] list, when check boxes of multiple base models for JMAG-RT are checked and [Create Model] is selected.

MEMO Two progress bars are displayed if two or more analyses are performed simultaneously. The upper progress bar indicates the progress of the analysis that is run currently. The lower progress bar indicates the progress of all analyses that are running on the machine.



When JMAG-RT completes all cases of the FEM analysis, the JMAG-RT model that is created is exported as a JMAG-RT model data file (*.rtt). The folder of the file can be confirmed in the [Information] tab opened by selecting a base model from the [RT Model Library] list, and then clicking [Modify]. The default folder is located in the following folder.

```
C: Mydocument\JMAG\JMAG_RT
```

Click [Setting] in the startup window, and then specify a new folder to change the folder.

A report file is displayed when the FEM analysis is complete regardless of whether the analysis is successful or failed. The cause of the failure is also displayed and can be confirmed in the report file.

MEMO The model type of a three-phase PM synchronous motor (LdLq or spatial harmonic) cannot be changed after a JMAG-RT model is created.

MEMO If the model type is a three-phase PM synchronous motor or linear synchronous motor, the motor characteristics, such as inductance and torque, base on the JMAG-RT model can be exported to a CSV file.

▶▶ Reference

See following pages for details.

- "13-8. Three-phase PM Synchronous Motor" on page 1036
- "13-11. Linear Synchronous Motor" on page 1104.

MEMO The basic information in the JMAG-RT model file can be checked using the JMAG-RT Viewer as well as the JMAG-RT Library Manager.

▶▶ Reference

For more information about the JMAG-RT Viewer, see "Chapter 14. JMAG-RT Viewer" on page 1187.

6. Copy the JMAG-RT model to a circuit/control simulator.

The JMAG-RT model data file exported in Step 5 is copied to the working directory of the circuit/control simulator. Furthermore, the files required to run a circuit/control analysis using a JMAG-RT model (MDL file, Netlist XML file, etc.) can be exported to the same directory.

Select the [Creation] check box of the base model that has [Created] displayed in the [Data Status] column of the [RT Model Library] list, and then click [Copy to Circuit/Control Simulator]. The [Copy to Circuit/Control Simulator] dialog box is displayed.

MEMO Multiple JMAG-RT models can be copied to a circuit/control simulator by selecting multiple JMAG-RT base models to copy to the working directory of the circuit/control simulator.

Specify the following settings in the [Copy to Circuit/Control Simulator] dialog box.

■ [Platform]

Specifies the circuit/control simulator to use.

Output files are varied by the specified circuit/control simulator.

- [Export RTT File]: Export JMAG-RT model data file (*.rtt) only. The file can be used by any platforms listed in above.
- [Simulink (electric signal output)]: Connects to the electric circuit of Simulink. SimPowerSystems of MATLAB/Simulink is required.
- [Simulink (voltage signal input)]: Inputs the control signal as voltage to each coil without connecting to the electric circuit of Simulink. The torque, electromagnetic force, and current flowing in each phase are output when [Simulink (voltage signal input] is selected.
- [Simulink (current signal input)]: Inputs the control signal as current to each coil. The torque and electromagnetic force are output when [Simulink (current signal input)] is selected.
- [PSIM]: Connects to the electric circuit of PSIM.
- [Other C Program]: Runs a simulation using a Program in C.

■ [Copy to]

Specifies the folder to run the analysis with a circuit/control simulator.

■ [Data File List]

Applies the name of the JCF file used in Step 1 to the JMAG-RT model that is created. Specify a new name in the [RTT File Name] (for Simulink) or [Netlist XML File Name] (for PSIM) column to change the name of the JMAG-RT model data file.

■ [Copy Subcircuit File]

Copies the following files with the JMAG-RT model data file.

- Simulink: MDl file and DLL file
- PSIM: Netlist XML file and DLL file

If [Copy Subcircuit File] is not selected, only the JMAG-RT model data file (*.rtt) is copied.

MEMO MDL file of Simulink
New MDL files are added to the three-phase PM synchronous motor for Simulink as of February 2010 (JMAG_RT_pmsm_FuvwLdLq.mdl, JMAG_RT_pmsm_Vin_FuvwLdLq.mdl, and JMAG_RT_pmsm_Iin_FuvwLdLq.mdl). The linkage flux for the U, V, and W phases and inductance for the d/q-axis are added to these new MDL files.

▶▶ Reference
For more information, see P.1054 "13-8-3. Circuit/Control Simulation with Simulink".

Click [OK] after specifying the settings.

MEMO A plot file, XML file containing the JMAG-RT settings (RTMOTOR.xml), a report file (HTML file), and an error file are exported when a JMAG-RT model is created (FEM analysis by JMAG-RT). Only the plot file and the XML file are copied for the JMAG-RT model to the circuit/control simulator.

7. Run the circuit/control simulation.

Arrange a JMAG-RT model block in the circuit of the circuit/control simulator.

Specify the parameters, such as coil resistance and moment of inertia, for the JMAG-RT model block, and then run the circuit/control simulation.

▶▶ Reference
See following chapters in details about each model. Notes for each plat-form are included.

- P.1036 "13-8. Three-phase PM Synchronous Motor"
- P.1073 "13-9. Two-phase Stepping Motor"
- P.1090 "13-10. Linear Solenoid"
- P.1104 "13-11. Linear Synchronous Motor"
- P.1126 "13-12. SR Motor"
- P.1139 "13-13. Induction Motor"

13-5. Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy)

Motor properties can be added to JMAG-RT models that are created.

An analysis can be run for new cases created by changing the parameters of a base model. The results can be added to the existing JMAG-RT model.

The analysis time can be reduced by calculating only the cases that have been added.

13-5-1. Important Notes Regarding the Use of This Function

- This is function does not support the induction motor.

13-5-2. Procedures for Executing this Function

The procedure for adding data to the JMAG-RT model is as follows.

1. **Select the base model that has [Created] displayed in the [Data Status] column of the [RT Model Library] list, and then click [Modify].**

   A message asking whether to unlock the data is displayed.

2. **Click [Yes].**

   A setting dialog for each motor is displayed

3. **Confirm the settings for the JMAG-RT model in the [Information] tab.**

   JMAG-RT model has already been created, information (current, angle, pole, etc) included in JMAG-RT model are displayed in [Basic Information] and [Resolution] group box in [Information] tab.

   MEMO Some of the data, such as the number of poles, may not be displayed in the [Information] tab depending on the version of JMAG-RT.

4. Modify the parameters.

The following parameters are supported in the [Parameter for RTT File] list. If parameters other than those listed in the table below (coil turns, etc.) the analysis may fail.

Model type	Parameter	
Three-phase PM synchronous motor	[Current (LdLq)]	[Simple Settings] • [Rated Current (A)] • [Current Phase Range (deg)] [Advanced Settings] • [Amplitude (A)] • [Current Phase (deg)]
	[Current (Spatial harmonic)]	[Simple Settings] • [Rated Current (A)] • [Current Phase Range (deg)] [Inductance] of [Advanced Settings] • [Amplitude (A)] • [Current Phase (deg)] • [Mechanical Angle Resolution (deg)] • [Input Current (A)] *In the case the drive type is [Other] [Torque] of [Advanced Settings] • [Amplitude (A)] * if [Use the Same Value for Inductance and Torque] is not selected • [Current Phase (deg)] * if [Use the Same Value for Inductance and Torque] is not selected • [Mechanical Angle Resolution (deg)] * if [Use the Same Value for Inductance and Torque] is not selected • [Periodic Angle (deg)] * if [Calculate Rotation Angle from the Number of Poles] is selected • [Input Current (A)] *In the case that the drive type is [Other] and [Use the Same Value for Inductance and Torque] check box is turned off.

Model type	Parameter	
Two-phase stepping motor	[Current]	• [Current Range (A)]
	[Resolution]	[Inductance] • [Division of Rotation Angle] • [Current Phase at Load (deg)] [Torque] • [Rotation Angle (deg)] • [Division of Rotation Angle] • [Current Phase at Load (deg)]
Linear solenoid	[Resolution (position)]	• [Number of Divisions]: A point sequence can also be created.
	[Resolution (current)]	• [Current Range (A)] • [Number of Divisions]: A point sequence can also be created.
Linear synchronous motor	[Current]	• [Current Amplitude Range (A)] • [Current Phase Range (deg)]
	[Resolution]	[Inductance] • [Number of Displacement Divisions] • [Current at Load (A)] [Electromagnetic Force] • [Displacement (m)] • [Number of Displacement Divisions] • [Current at Load (A)] • [Current Phase at Load (A)]

Model type	Parameter	
SR Motor	[Current]	[Simple Settings] • [Rated Current (A)] [Inductance] of [Advanced Settings] • [Amplitude (A)] • [Mechanical Angle Resolution (deg)] [Torque] of [Advanced Settings] • [Amplitude (A)] * if [Use the Same Value for Inductance and Torque] is not selected • [Mechanical Angle Resolution (deg)] * if [Use the Same Value for Inductance and Torque] is not selected • [Periodic Angle (deg)] * if [Calculate Rotation Angle from the Number of Poles] is selected

5. Select [Recalculate with Increased Resolution] from the [Calculation Mode] list in the [Create] tab.

If [Redo Only Failed Cases] is selected, the JCF and plot files are checked for any failed cases, and the case(s) that have failed are re-calculated.

If [Redo All Calculations] is selected, the JMAG-RT model data file, the JCF file, and the plot file for each case is deleted, and the analysis is re-calculated.

6. Click [OK] to return to the JMAG-RT Library Manager.

The startup window is displayed.

7. Create the JMAG-RT model (run the FEM analysis).

An FEM analysis is run for only the cases that have been added.

The JMAG-RT model data file before the re-calculation is re-named and saved to the same directory. For instance, if the original JMAG-RT model data file is “XXX.rtt”, it is re-named as follows.

XXX-20060530-134715.rtt (Original file name - date - hhmmss.rtt)

■ Calculating cases that are added

For example, a JMAG-RT model is created by specifying the following current amplitude for a three-phase PM synchronous motor.

```
0, 0.5, 1, 1.5, 2
```

The JMAG-RT model is created again by modifying the current amplitude as follows after a JMAG-RT model has been created.

```
0, 1, 2, 3, 4
```

The analysis has been run for the current values of 0, 1, and 2, therefore, the analysis is only run for the current values of 3 and 4.

The results for the current values of the JMAG-RT model include the following current amplitudes.

```
0, 0.5, 1, 1.5, 2, 3, 4
```

13-6. Creating JMAG-RT Model from a CSV File

JMAG-RT model file can be created from a CSV file as well as from an FEM model (*.jcf) created by JMAG. The CSV file can contain user prepared data or the result of actual measurements.

▶▶ Reference

See P.1021 "13-4. Using JMAG-RT (JMAG-RT Library Manager)"about the operation of CSV file import.

13-6-1. Important Notes Regarding the Use of This Function

Note following points when using this function.

- Three phase PM synchronous motor is supported.
- A JMAG-RT Ld, Lq model is created using this function. The Spatial Harmonic Model is not supported.
- The JMAG-RT model file includes an inductance table whose axes are amplitude and phase. Hence setting resolution is necessary to expand data in Id and Iq axes to amplitude and phase axes.
- Ironloss cannot be included.

▶▶ Reference

See also P.1038 "13-8-2. Settings in the JMAG-RT Library Manager".

13-6-2. Input File Format

CSV file format is below.

```
Line 1: Flux
Line 2: Magnetic flux [Wb]
Line 3: Colum1: Id, Comun2: Iq, Column3: Ld, Column4: Lq
Line 4 and following lines: Id, Iq, Ld, Lq for each column.
Unit is [H]
```

Flux
1.23400
Id, Iq, Ld, Lq
0.000000, 0.122447, 1.000000, 31.000000
-0.106607, 0.061224, 2.000000, 32.000000
-0.122447, 0.000000, 3.000000, 33.000000
-0.106607, -0.061224, 5.000000, 35.000000
-0.061224, -0.106607, 6.000000, 36.000000

Example of CSV file

MEMO Line1 and Line3 are fixed strings.

13-7. Updating JMAG-RT Model Data File Created by Previous Version

The file format of JMAG-RT model data file is modified from JMAG-Designer 10.5 and JMAG-Studio 10.03001 or later.

The JMAG-RT Viewer does not support an old version of a JMAG-RT model data file, in which it is necessary to convert the file format to the new version in order to display a JMAG-RT model data file in JMAG-RT Viewer.

▶▶ Reference

For more information about JMAG-RT Viewer, see "Chapter 14. JMAG-RT Viewer" on page 1187.

13-7-1. Important Notes Regarding the Use of This Function

- A set of circuit/control simulation files written before JMAG-Designer 10.5 and MAG-Studio 10.03001 cannot be used for the updated JMAG-RT data files.
  It is necessary to use a set of files for the circuit/control simulation written from JMAG-Designer 10.5 or JMAG-Studio 10.03001 or later when using the updated JMAG-RT data files.
- The [Creation Date] displayed in the [Data Information] tab is the date which the JMAG-RT file was updated.

13-7-2. Operation for JMAG-RT Data File Conversion

The following procedures are for converting JMAG-RT data files.

1. **Start the JMAG-RT Library Manager bundled in JMAG-Designer 10.5 or JMAG-Studio 10.03001 or later.**
2. **Select a JMAG-RT motor model from the [RT Model Library] list in the main dialog that is set to [Created] in the [Data Status] column and click [Modify].**

   A message asking if you want to edit the current data is displayed.

3. **Select [No] in the dialog box and dialog box to set each model type is displayed.**
4. **Confirm the information for the JMAG-RT model file displayed in the [Data Information] tab.**
5. **Select [OK] and a message asking if you want to update the JMAG-RT model data file is displayed.**
6. **Select [Yes] and a report file is displayed on the screen whether or not updating the data has succeeded or not.**

   Confirm the error message that is displayed explaining the error if it occurs.
7. **The JMAG-RT model is copied to the working folder for the circuit/control simulator.**

   The JMAG-RT model data file that is created to the working folder is converted to the new format.

13-8. Three-phase PM Synchronous Motor

13-8-1. Creating an FEM Model with JMAG

An FEM model needs to be created to meet the following requirements.

- Create a FEM model for a transient response analysis (TR, DP).
- Three FEM coil conditions need to be specified.
  The FEM model cannot be imported into JMAG-RT if there are more or less than three FEM coil conditions.
- A rotation motion condition needs to be specified.
  - Select [Rotation] under [Type] in the [Motion] dialog box for JMAG-Studio. Furthermore, select [Constant] for [Method] under [Displacement].
  - Select [Constant Revolution Speed] in the [Rotation Motion] settings for JMAG-Designer.

  JMAG-RT uses the FEM model settings described above even if the settings that are specified differ.
- The initial position of the rotor can be set to any position. JMAG-RT automatically determines the position for the current phase at zero based on the flux linkage calculation when creating a JMAG-RT model.
- A torque condition needs to be specified to create a spatial harmonics model.
  - Specify an electromagnetic force and torque calculation condition, and then select the [Calculate Torque] check box when creating an FEM model in JMAG-Studio.
  - Specify a torque condition, such as nodal force, Lorentz force, or surface force, when creating an FEM model in JMAG-Designer.

  A torque condition is not required for an Ld, Lq model.

▶▶ Reference
For more information about creating a JMAG-RT model for a three-phase PM synchronous motor, see P.1038 "13-8-2. Settings in the JMAG-RT Library Manager"

- Setting an Ironloss condition in the JCF file is needed when considering the ironloss effect.
- The conversion factors for both the results and the circuit need to be specified if a partial model is used.
  - Specify the conversion factors in the [Analysis Control] dialog box for JMAG-Studio.
  - Specify the conversion factors in the [Study Properties] dialog box for JMAG-Designer.

  JMAG-RT uses the results obtained from an FEM analysis as full model values.

  The following should be noted.
- The FEM coil condition needs to be associated with an FEM coil component by creating a circuit in a standard FEM analysis, but a circuit and the FEM coil components do not need to be specified to use JMAG-RT.

13-8-2. Settings in the JMAG-RT Library Manager

Two types of JMAG-RT models can be created for a three-phase PM synchronous motor.

MEMO The type of JMAG-RT model can be selected at [Select Motor Model] dialog (See step3 at "13-4. Using JMAG-RT (JMAG-RT Library Manager)" on page 1021).
This dialog is dis-played when creating a new motor model and clicking the [Change Model] button in the [Create] tab.

■ Ld,Lq Model

The Ld,Lq model has Ld/Lq current and a phase angle dependency. The effects of harmonics due to the slots cannot be simulated because only a sinusoidal component of electromotive force for the motor is used to calculate Ld and Lq.

MEMO An Ld,Lq model is created from user prepared data (*.csv).

Important **Turn on [Ld, Lq model] check box to enable the display of torque-speed curve or efficiency map in the JMAG-RT Viewer.**

▶▶ Reference
For more information about the JMAG-RT Viewer, see "Chapter 14. JMAG-RT Viewer" on page 1187.

■ Spatial Harmonic Model

The spatial harmonic model has inductance, torque with current, and a phase angle dependency. The effects of harmonics due to the slots can be simulated. The inductance can be obtained highly accurately even if the model has a very high magnetic saturation.

MEMO The Ld,Lq model and spatial harmonic models employ different method of calculation. Hence calculation time when checking [Ld,Lq Model] and [Spatial Harmonic Model] is a summation of calculation time for each [Ld,Lq Model] and [Spatial Harmonic Model].

MEMO The Simple Spatial Harmonic Model supported by JMAG-Designer 10.4 and JMAG-Studio 10.02801 or older versions are integrated into the Spatial Harmonic Model. Setting for the Simple Spatial Harmonic Model in old JMAG-RT model can be displayed in the latest JMAG-RT Library Manager, but Simple Spatial Harmonic Model type can not be selected when creating new motor model.

13-8-2-1. [Information] Tab

The [Information] tab manages information in the JMAG-RT model.

■ [Basic Information]

The following items can be set.

- [Title]
- [Created By]

The following items are set automatically after the model is created by JMAG-RT.

- [File Name]
- [Motor Type]
- [Creation Data]
- [JMAG Version]
- [RTT File Version]
- [Model Type]

■ [Motor Information]

The following items can be set.

- [Weight [kg]]
- [Shaft Length [m]]
- [External Diameter [m]]
- [Moment of Inertia [kgm^2]]
- [Rated Speed [rpm]]
- [Rated Power [kW]]
- [Rated Current [A]]
- [Rated Voltage [V]]

Important **These items do not affect the model creation process of JMAG-RT.**

The following items are set from the values in the [Creation] tab.

- [Number of Poles]
- [Resistance [ohm]]

The following items are set automatically after the model is created by JMAG-RT.

- [Permanent Magnet Flux [Wb]]

- [Ld [H]], [Lq [H]]: Average values are displayed. These values are used for circuit/control simulation when [LdLq (Constant)] option is selected in the parameter [Accuracy Type] of the JMAG-RT model block in Simulink.
  Turn on [Publish] check box to enable the display of these items in the JMAG-RT Viewer.

■ [Comment]

Any text can be entered here and will be stored with the JMAG-RT model. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Resolution]

Each item is set automatically after the model is created by JMAG-RT. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Model Folder]

The JMAG-RT model data file (*.rtt) is exported to the folder indicated in this text box. The HTML file (JCF file name.html) created when the JMAG-RT model data is created is exported to this folder with the model data.
Furthermore, a "run" folder is created in this folder when a JMAG-RT model is created. The PLOT file, etc., of the FEM analysis is exported to the run folder.

13-8-2-2. [Create] Tab

The [Create] tab specifies the parameters for the FEM analysis required to create a JMAG-RT model.

■ [Motor Model]

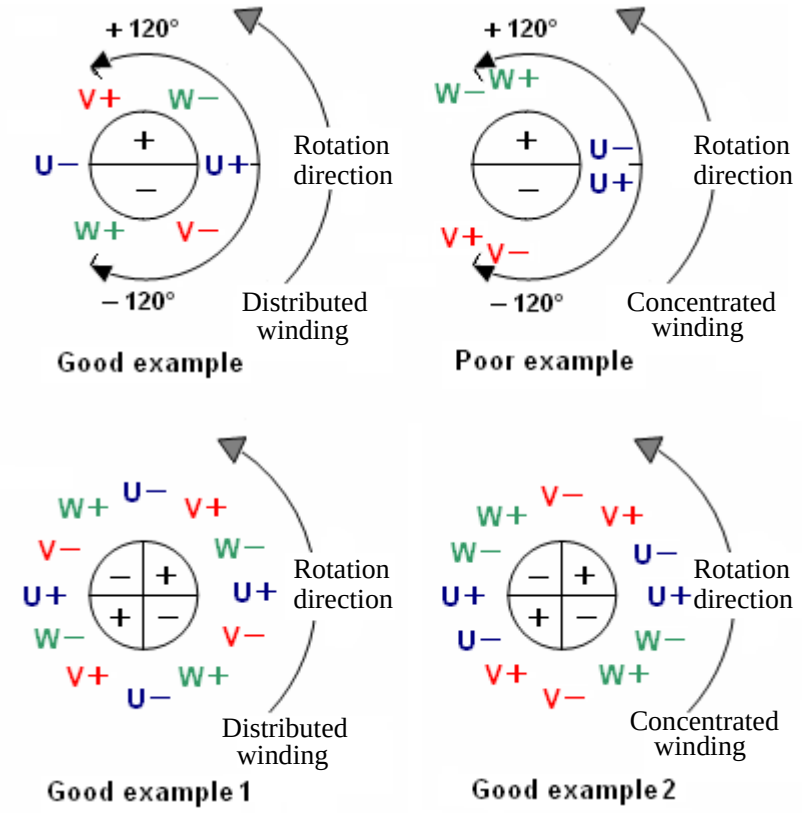
Changes the model type. Click [Change Model] to open the [Select Motor Model] dialog box. However, the model type cannot be changed once a JMAG-RT model has been created.

■ [Number of Poles]

Specifies the number of poles for the motor.

■ [Configuration of FEM Coil]

- [U-Phase], [V-Phase], [W-Phase]: Displays the FEM coil conditions set for the FEM model. Specify the condition is associated with each phase. The motor rotates counterclockwise and the phase difference between the three phases is defined relative to the U-phase. V-phase is displaced by the electric angle of -120 degrees and the W-phase by +120 degrees.



- [Number of Turns]: Specifies the number of turns for the coil.
- [Connection Pattern]: Specifies [Y-connection] or [Delta-connection].
- [Resistance (ohm)]: Specifies the phase resistance of the coil. The phase resistance of the coil is used as the default value for the MagCoupler-RT PMSM Machine block in PSIM. [Resistance (ohm)] is not used for Simulink.

■ [Current (Ld, Lq)]

Important **Specifies the current including the current flowing while driving the motor. If the upper limit of current is not large enough, the effect of magnetic saturation may not be taken into account.**

MEMO If the current exceeds the specified range when using the JMAG-RT model in a circuit/control simulator, the inductance and torque values obtained at the maximum or minimum current are applied to the model instead of extrapolating the data points.

To create Ld,Lq model the following items should be specified.

- [Simple Settings]: This setting causes the number of points in the JMAG-RT model to be set automatically from the range of rated current and current phase.
  The following items are specified for this setting.
  - [Rated Current (A)]
  - [Current Phase Range (deg)]

  MEMO The JMAG-RT model that is created has a range of current between 0 A and three times the specified rated current. Seven data points are specified for the current.

  MEMO The current phase has data points every 10 degrees for the specified range and every 45 degrees beyond that range.

- [Advanced Settings]: This setting allows the current sequence data to be set directly. Click [Setting] button then the [Advanced Settings of Current] dialog is displayed.
  The following items are specified for this setting.
  - [Amplitude (A)] : Click [Point Sequence] and specify a point sequence of current amplitude.
  - [Current Phase (deg)] :Click [Point Sequence] and specify a point sequence of current phase.

  MEMO The [Create Point Sequence] dialog box is displayed by clicking [Create Point Sequence] in the [Current Point Sequence] dialog box. Specify [Min], [Max], and [Division], and then click [OK]. The point sequence that is created is displayed in each point sequence dialog box.

  It is necessary to specify the following items to create a JMAG-RT model from a CSV file.
  - [Resolution Point Sequence]: A resolution is specified to expand Id, Iq axes data into amplitude, phase axes data.

■ [Current (Spatial Harmonics)]

Important **Specifies the current including the current flowing while driving the motor. If the upper limit of current is not large enough, the effect of magnetic saturation may not be taken into account.**

MEMO If the current exceeds the specified range when using the JMAG-RT model in a circuit/control simulator, the inductance and torque values obtained at the maximum or minimum current are applied to the model instead of extrapolating the data points.

- [Drive Type]: Select the type of current waveform of the motor to drive in the circuit/control simulator according to whether the motor has a sinusoidal or non-sinusoidal current waveform.

  Select from following items as the drive type.

  - [Sine Wave Drive]: Select [Sine Wave Drive] for a current applied to the motor that has a sinusoidal waveform.
  - [Other]: Select [Other] when the current waveform is not sinusoidal.

  MEMO The time required to create a JMAG-RT model varies depending on the current waveform pattern.

  MEMO The JMAG-RT model contains the rotation angle and current dependent, inductance, and torque data.

Select [Simple Settings] or [Advanced Settings] for the current settings.

- [Simple Settings]:

  In the case that the [Drive Type] is [Sine Wave Drive], the number of data points in JMAG-RT model is specified automatically from the range of rated current and current phase.

  - [Rated Current (A)]
  - [Current Phase Range (deg)]

  In the case that [Drive Type] is [Other], the number of data points in JMAG-RT model is specified automatically from the rated current.

  - [Rated Current (A)]

  MEMO The current values of all the possible combinations specified for each rotation angle are applied to each coil. The JMAG-RT model contains the data for these current values.

  MEMO The JMAG-RT model that is created has a range of current between 0 A and three times the specified rated current. Seven data points are specified for the current.

MEMO The current phase has data points every 10 degrees for the specified range and every 45 degrees beyond that range.

- [Advanced Settings]

  In the case that the [Drive Type] is [Sine Wave Drive], this setting specifies the dependency of current amplitude and current phase using point sequence.
  Click [Setting] then [Advanced Settings of Current] is displayed.

  Specify the following items for the inductance setting.

  - [1/2 Electric angle with Periodicity]: If the angle dependency of the inductance for each phase has a periodicity of 1/2 electric period, select the [1/2 Electric Angle with Periodicity] check box. The calculation time to create a JMAG-RT model is reduced.
  - [Amplitude (A)]: Click [Point Sequence] then specify a point sequence of current amplitude.
  - [Current Phase (deg)]: Click [Point Sequence] then specify a point sequence of current phase.
  - [Mechanical Angle Resolution (deg)]

  MEMO The [Create Point Sequence] dialog box is displayed by clicking [Create Point Sequence] in the [Current Point Sequence] dialog box. Specify [Min], [Max], and [Division], and then click [OK]. The point sequence that is created is displayed in each point sequence dialog box.

  Specify following items for the torque setting.

  - [Use the Same Value for Inductance and Torque]: If the same data points can be defined for both inductance and torque, select the [Use the Same Value for Inductance and Torque] check box. The calculation time to create a JMAG-RT model is reduced.
  - [Amplitude (A)]: Click [Point Sequence] then specify a point sequence of current amplitude.
  - [Current Phase (deg)]: Click [Point Sequence] then specify a point sequence of current phase.
  - [Mechanical Angle Resolution (deg)]
  - [Periodic Angle (deg)]: Input a range of a cycle of torque ripple.
  - [Calculate Rotation Angle from the Number of Poles]: If the periodicity of torque ripple can be determined from the number of poles, select the [Calculate Rotation Angle from the Number of Poles] check box. The angle per period is automatically calculated.

MEMO The [Create Point Sequence] dialog box is displayed by clicking [Create Point Sequence] in the [Current Point Sequence] dialog box. Specify [Min], [Max], and [Division], and then click [OK]. The point sequence that is created is displayed in each point sequence dialog box.

In the case that the [Drive Type] is [Other], the dependency between inductance and torque can be specified using a point sequence. The [Advanced Setting of Current] dialog box is displayed by clicking the [Setting] button.

Specify the following items for the inductance setting.

- [Input Current (A)]: Click [Point Sequence] button then specify a point sequence for the current.
- [Mechanical Angle Resolution (deg)]

MEMO The [Create Point Sequence] dialog box is displayed by clicking [Create Point Sequence] in the [Current Point Sequence] dialog box. Specify [Min], [Max], and [Division], and then click [OK]. The point sequence that is created is displayed in each point sequence dialog box.

Specify following items for the torque setting.

- [Use the Same Value for Inductance and Torque]: If the same data points can be defined for both inductance and torque, select the [Use the Same Value for Inductance and Torque] check box. The calculation time to create a JMAG-RT model is reduced.
- [Input Current (A)]: Click [Point Sequence] button then specify a point sequence for the current.
- [Mechanical Angle Resolution (deg)]
- [Periodic Angle (deg)]: Input a range of a cycle of torque ripple.
- [Calculate Rotation Angle from the Number of Poles]: If the periodicity of torque ripple can be determined from the number of poles, select the [Calculate Rotation Angle from the Number of Poles] check box. The angle per period is automatically calculated.

MEMO The [Create Point Sequence] dialog box is displayed by clicking [Create Point Sequence] in the [Current Point Sequence] dialog box. Specify [Min], [Max], and [Division], and then click [OK]. The point sequence that is created is displayed in each point sequence dialog box.

MEMO The current values of all the possible combinations specified for each rotation angle are applied to each coil. The JMAG-RT model contains the data for these current values.

■ [Iron Loss]

When considering the iron loss effect, turn on the [Allow Iron Loss] check box and specify the following items.

- [Simple Settings]: Specify [Maximum Revolution Speed (rpm)]. The model calculation process will use speed increments fifth part of the maximum revolution speed. For example, when the [Maximum Revolution Speed (rpm)] is set to 3600rpm, the model creation process will use revolution speed of 0, 720, 1440, 2160, 2880 and 3600.
- [Advanced Settings]: Click the [Point Sequence] button then the [Point Sequence] dialog is displayed. Specify a point sequence for the revolution speeds.
  When clicking the [Create Point Sequence] button, the [Create Point Sequence] dialog is dis-played. Specifying [Min], [Max] and [Division] then a point sequence for the revolution speeds will be created from the min, max and division value. And the created point sequence is set in the [Point Sequence] dialog.

  MEMO Circuit/Control simulators refer to the iron loss table using an absolute value of the revolution speed hence setting negative value as the revolution speed is allowed. But it is assumed that the iron loss table is created for normal rotation with positive speed, so that it may not match the value which is correct physically.

  Important **Iron loss cannot be considered coupling with PSIM.**

  Important **Turn on the [Allow Iron Loss] check box if it is required to display the torque-speed curve or efficiency map considering the iron loss effect with the JMAG-RT viewer.**

  ▶▶ Reference
  For more information about the JMAG-RT Viewer, see "Chapter 14. JMAG-RT Viewer" on page 1187.

■ [Distributed Calculation]

Select the [Use Distributed Calculation] check box when distributing the calculation to multiple machines on the same network using JMAG-RT, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

Important **When using this function, it is required to set up JMAG-Remote System.**

■ [Calculation Mode]

Specifies the type of calculation to re-calculate the data that has already been calculated from the following.

- [Recalculate Only Failed Cases] : Re-calculates only the failed cases.

  The JMAG-RT model data file (*.rtt) cannot be exported until after all the cases of an analysis have been completed in JMAG-RT. The calculation time is reduced by calculating only the failed cases.

  Important **When re-calculating the failed cases, make sure to change the run folder in which the PLOT file of the FEM analysis is saved. The run folder can be created in the model folder of the [Information] tab.**

- [Recalculate with Increased Resolution]: Calculates only the cases that are added and adds the results to the existing JMAG model data file if a part of the data that has already been analyzed is modified.

  ▶▶ Reference

  For more information about adding data, see P.1028 "13-5. Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy)".

- [Redo All Calculations]: Re-calculates all of the cases even if there is only one case that fails. If there are parameters that cannot be added to a JMAG-RT model data file (*.rtt), all the cases need to be re-calculated.

■ [Do Not Output Distribution to Result File]

Specifies to output only the necessary data to the plot file when creating a JMAG-RT model ignoring the settings specified in the JCF file. If the [Do Not Output Distribution to Result File] is not selected, the output control settings specified in the JCF file are used for creating a JMAG-RT model. Note that this requires a large amount of memory.

MEMO In the case that the [Allow Iron Loss] check box is turned on, the distribution of the magnetic flux density will be output irrespective of whether the [Do not Output Distribution to Result File] is turned on or not.

■ [Do not leave result files]

The [Do not leave result files] is selected when running a new calculation for JMAG-RT that automatically deletes all the plot files in the working folder.

■ [Delete result file]

The plot files that are exported during calculation are deleted when a JMAG-RT model folder is already created and the [Delete result file] button is selected. The [Delete result file] button can be selected when result files that are written during the calculation exist in the run folder.

13-8-2-3. [Motion] Tab

■ [Equation of Motion (PSIM only)]

Specifies the moment of inertia and time constant for a JMAG-RT model to use in the PSIM control/circuit simulator. These values are used as default values in the MagCoupler RT PMSM Machine block.

The JMAG-RT model created for Simulink contains these values, but the settings cannot be used in Simulink. These parameters need to be specified again in Simulink.

A correction coefficient can be applied to the moment of inertia. The correction coefficient is as follows.

$$J = \mathrm{coef\_inertia\ 1} \cdot J_0 + \mathrm{coef\_inertia\ 2}$$

$$J\frac{d\omega}{dt} = T - T_{load} - B \cdot \omega$$

$$B = \frac{J}{\tau_{mech}}$$

- [Inertia Moment J_0 (kg*m^2)]: Specifies the inertia moment.
  - The value specified for [Inertia Moment for Target Region J1] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Inertia J_1] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.
- [Inertia Moment Correction Coefficient *(coef_inertia 1)*]: Specifies a correction coefficient for the inertia moment.
  - The value specified for [Coefficient p] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Inertia Coefficient, p] in the [Define Equation of Motion] dialog box in JMAG-Designer is displayed by default.
- [Inertia Moment Correction Coefficient *(coef_inertia 2)*]: Specifies a correction coefficient for the inertia moment.
  - The value specified for [Correction Value J_2] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Correction Value J_2] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.
- [Time Constant τ_{mech} (sec)]: Specifies a time constant.

13-8-2-4. [RT Correction] Tab

The [RT Correction] tab is displayed after a JMAG-RT model is created (after an FEM analysis using JMAG-RT). The correction coefficient can be specified for each parameter exported to the JMAG-RT model data file.

MEMO The correction coefficients can also be specified in a control/circuit simulator.

MEMO The specified correction coefficients are cleared and default values are used to re-create a JMAG-RT model.

■ [Temperature Correction]

The temperature correction is for both the magnet and the coil. The correction equation is as follows. The correction equation is as follows.

Correction equation for magnet

$$F' = F\left(1 + \frac{k}{100}(Tf_{now} - Tf_{base})\right)$$

F': Magnetic flux after correction (Wb)

F: Magnetic flux before correction (Wb)

K: Temperature correction coefficient of magnet (%/K)

The range of the coefficient is -100.0 - 0.0 (negative)

Default: -0.11%/K

Tf_{now}: Reference temperature for magnet (K)

Tf_{base}: Base temperature for magnet (K)

Correction equation for coil

$$R' = R(1 + \alpha(Tr_{now} - Tr_{base}))$$

R': Resistance after correction (ohm)

R: Resistance before correction (ohm)

α: Temperature correction coefficient of coil (ppm/K)

The range of the coefficient is 0.0 and over

Default value is 3810 ppm/K

Tr_{now}: Reference temperature of coil (K)

Tr_{base}: Base temperature of coil (K)

Default: 293.15K

Specify the base temperature and correction coefficient for the magnet and the coil appeared in the above equation.

Important **In the case that a temperature terminal of the S-Function block has no connection, the reference temperature is treated as zero for the calculation.**

Important **The temperature correction function cannot be used with PSIM.**

■ [Direct Correction], [Complex Correction], [Reference Correction]

The correction equation is as follows.

Correction equation for inductance: $L_{new} = K_1 \times K_5 \times \frac{K_6}{n} \times L(I_{new})$

Correction equation for linkage flux:

$$\Psi_{new} = K_2 \times K_4 \times K_5 \times \frac{K_6}{n} \times \frac{K_7}{n} \times \Psi$$

Correction equation for torque: $T_{new} = K_3 \times K_4 \times K_5^2 \times \frac{K_6}{n} \times T(I_{new})$

Correction equation for current: $I_{new} = \frac{K_7}{n} \times I$

L_{new} : Inductance after correction (H)

Ψ_{new} : Linkage flux after correction (Wb)

T_{new}: Torque after correction (Nm)

I_{new} : Reference current value after correction (A)

* Used when referring to the inductance and torque.

L : Inductance before correction (H)

Ψ : Linkage flux before correction (Wb)

T : Torque before correction (Nm)

I: Reference current value before correction (A)

n: Number of turns (turn)

Specify the following correction coefficients based on the equations above.

■ [Direct Correction]

- Inductance (coef_inductance) K_1
- Flux Linkage in Magnet (coef_flux) K_2
- Torque Correction (coef_torque) K_3

- Current Dependent Torque Point Sequence (coeftable_torque) K_{pa3}

  The current dependent point sequence is used to correct values according to the current amplitude. Select the [Use Current-Dependent Table for Torque Correction (coeftable_torque_flag)] check box, click [Point Sequence], and then specify the point sequence.
  The following should be noted when the current dependent point sequence is used.

  - If the current exceeds the specified range when using the JMAG-RT model in a control/circuit simulator, the inductance and torque values obtained at the maximum and minimum currents are applied instead of extrapolating.
  - The point sequence can be created by reading the data from the point sequence file (*.pa). Click [Read] in the [Point Sequence] dialog box, and then specify the point sequence file.

```
Magnet

                0                   5
    0.000000000000e+00      0.000000000000e-00
    5.000000000000e-01      0.500000000000e+03
    1.000000000000e+00      0.500000000000e+05
    0.000000000000e+00      0.000000000000e-00
    5.000000000000e-01      0.500000000000e+03
```

Sample of a point sequence file

  - The correction point sequence data is exported to a point sequence file (*.pa) in addition to the JMAG-RT model data file (*.rtt) when the JMAG-RT model is copied to a control/circuit simulator. The same file name as the JMAG-RT model data file is used for this point sequence file.
    The name of the point sequence file is used in the JMAG-RT model data file and cannot be changed. Furthermore, the JMAG-RT model data file and point sequence file need to be stored in the same directory.

■ [Complex Correction]

- Magnet (coef_magnet) K_4
- Steel Sheet (coef_material) K_5
- Number of Turns (turns_coil1) K_6 (Turn)

■ [Reference Correction]

- Current Reference (coilturns_coil2) K_7 (Turn)

13-8-2-5. [Motor Characteristics] Tab

The [Motor Characteristics] tab is displayed after a JMAG-RT model is created (after FEM analysis using JMAG-RT). Characteristics, such as torque and inductance, can be exported from the JMAG-RT model that is created in the CSV file format.

MEMO The correction coefficient specified in the [RT Correction] tab is not taken into account in the table data that is exported.

■ [Output Items]

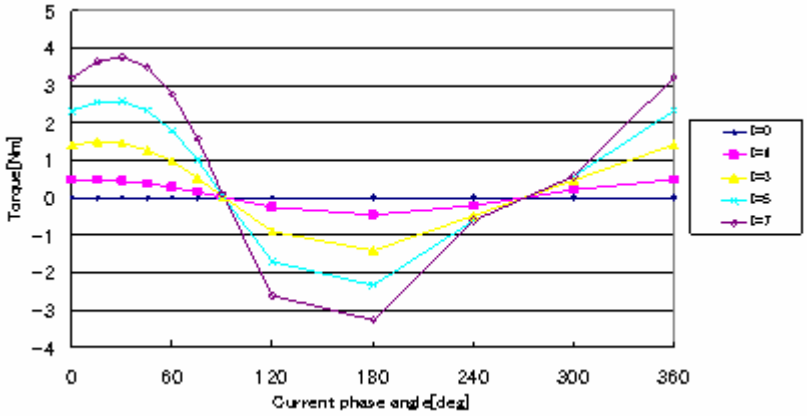
Turn on the following check box and click the [Export] button.

- [Inductance (Lp, Lq)]: Exports the following CSV file. Select the [Inductance (lp, Lq)] check box, and then click [Export].
  - Ld_graph.csv, Lq_graph.csv: Contains inductance for each rotation angle (mechanical angle) at the current phases of the d-axis and q-axis.
- [Torque]: Exports the following CSV file. Select the [Torque] check box, specify a value in the [Current Phase (deg)] box, and then click [Export].
  - AveTrq_graph.csv: Contains average torque for each current phase at each current value calculated with JMAG-RT.
  - Trq_graph.csv: The torque of the rotation angle (mechanical angle) dependent with each current value at the angle specified in [Current phase (deg)].
    Furthermore, the points specified for the number of divisions for the torque rotation angle is exported when a JMAG-RT model data is created.

    MEMO [Current phase (deg)] indicates the phase of current.

MEMO [Current phase (deg)] is not available for an Ld, Lq model.

Beta[deg]	I=0.000000	I=1.000000	I=3.000000	I=5.000000	I=7.000000
0	-0.000057	0.47098	1.403186	2.319068	3.207819
15	-0.000057	0.472006	1.484304	2.549191	3.6301
30	-0.000057	0.438146	1.444567	2.566549	3.752475
45	-0.000057	0.369105	1.267454	2.322011	3.484685
60	-0.000057	0.268923	0.95495	1.794876	2.75523
75	-0.000057	0.145411	0.527851	1.014182	1.580275
90	-0.000057	0.008845	0.032564	0.063134	0.09954
105	-0.000057	-0.128501	-0.467352	-0.899729	-1.404698
120	-0.000057	-0.254345	-0.905989	-1.70778	-2.628106
135	-0.000057	-0.357918	-1.234485	-2.270585	-3.418104
150	-0.000057	-0.430985	-1.429761	-2.550944	-3.741786
165	-0.000057	-0.469628	-1.486507	-2.56479	-3.664056
180	-0.000057	-0.473016	-1.419433	-2.358312	-3.275017



- [Induced Voltage]: Exports the following CSV file. Select the [Induced Voltage] check box, specify a value in the [Rotation Speed (rpm)] box, and then click [Export].
  - InducedVoltage_graph.csv: Contains the induced voltage with no-load at the specified number of rotations.

13-8-3. Circuit/Control Simulation with Simulink

■ Signal Types

There are three types of signal inputs in Simulink; electric signal input, voltage signal input, and current signal input.

- A JMAG-RT model (electric signal input) can be used by connecting to the electric circuit of Simulink.
- A JMAG-RT model (voltage signal input) can be used to input the voltage signal to each coil, and then output the torque or current flowing in each phase of coils without connecting an electric circuit.
- A JMAG-RT model (current signal input) can be used to input the current signal to each coil, and then output torque without connecting an electric circuit.

MEMO The SimPowerSystems of MATLAB/Simulink is not required when a JMAG-RT model (voltage signal input or current signal input) is used.

■ Related Files

The files required for a simulation with Simulink are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor.
RT_simulink.dll	Required to use a JMAG-RT model in Simulink.
JMAG_RT_pmsm.mdl	Specifies a three-phase PM synchronous motor for the JMAG-RT model (electric signal input). This file is exported when [Simulink (electric signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.

File name	Description
JMAG_RT_pmsm_Vin.mdl	Specifies a three-phase PM synchronous motor for the JMAG-RT model (voltage signal input). This file is exported when [Simulink (voltage signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.
JMAG_RT_pmsm_Iin.mdl	Specifies a three-phase PM synchronous motor for the JMAG-RT model (current signal input). This file is exported when [Simulink (current signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.
Torque correction point sequence file (*.pa)	Copied when the correction point sequence for the current dependent torque is specified in the [RT Correction] tab of JMAG-RT Library Manager after a JMAG-RT model is created (after the FEM analysis using JMAG-RT). The same file name as the JMAG-RT model data file (*.rtt) is used. *This file name is used in the JMAG-RT model data file (*.rtt) and cannot be changed.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
- These files must be in the same directory when a circuit/control simulation is run.
- Only the files exported from the same version of JMAG-RT can be used for a circuit / control simulation. All of the files listed in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

A MDL block needs to be arrange in the circuit of Simulink. A MDL block in a MDL file can be opened in the Command Window.

■ Properties of JMAG-RT model block

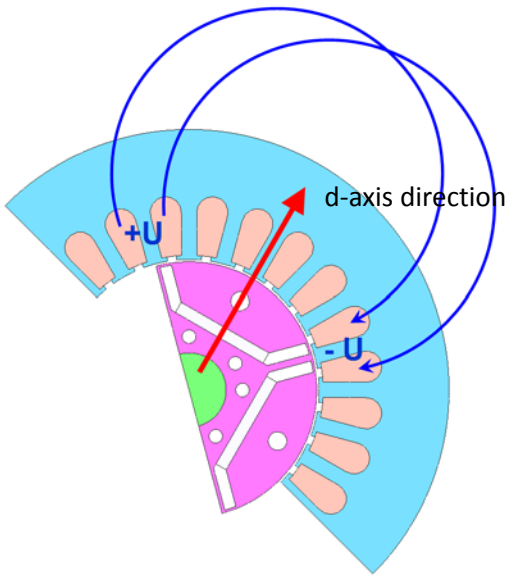
The following settings can be specified for the properties of a JMAG-RT model block.

MEMO If a correction coefficient is specified in the JMAG-RT Library Manager, the value is used as the default value in Simulink.

Parameter	Description	Signal type
RT data file	Specifies a name for the JMAG-RT data file.	Electric signal input, voltage signal input, current signal input
Accuracy Type	Specifies a model type. There are four types of model as described below. • LdLq(Table) The result of the Ld Lq model is used. • Simple Spatial Harmonic The result of the Simple Harmonic model is used • Spatial Harmonic The result of the Spatial Harmonic model is used. • LdLq(Constant) The result of the Ld Lq model is used. Instead of the table values, an averaged Ld Lq value are applied. The averaged Ld Lq value will be displayed in the [Information] tab of the [Three phase PM Synchronous Motor] dialog. When an invalid model type is specified, an error will occur during the calculation.	
Use IronLoss	Flag for using iron loss Results for the iron loss can be obtained if a JMAG-RT model data file with iron loss information is used to run a calculation and the [Use IronLoss] check box is selected. Iron loss information is not taken into account when the [Use IronLoss] check box is selected and a JMAG-RT model data file without iron loss information is used.	
Use pwm source	Flag for using pwm signal Select this setting when using the iron loss in a circuit using pwm signal. The calculation may be incorrect when this setting is not specified.	

Parameter	Description	Signal type
pwm frequency	Frequency for pwm signal carrier frequency Specify a carrier frequency for a pwm signal when using an iron loss in a circuit that is using a pwm signal. The calculation may be incorrect when this item is not specified.	Electric signal input, voltage signal input, current signal input
Moment of inertia ($Kg*m^2$)	Specifies the inertia moment ($Kg*m^2$) of the rotating machine. The value specified in the JMAG-RT Library Manager is not used.	
Friction factor (Nm/(rad/s))	Specifies the friction coefficient (Nm/(rad/s)). The value specified in JMAG-RT Library Manager is not used.	
Resistance of coil (ohm)	Specifies the resistance of the coil (ohm). The value specified in the JMAG-RT Library Manager is not used.	Electric signal input, voltage signal input
Factor	Specifies the calculation accuracy. The default value is '1000'. If the accuracy cannot be improved by increasing the time step, increase the number of digits by one. Make sure to avoid specifying a value smaller than the default value. The calculation cannot achieve high accuracy if the factor that is specified is too small.	Electric signal input
Sample Time	Specifies the time step for the analysis. If the default value of "-1" is used, the time step specified in Simulink is used in JMAG-RT. If it takes too long to calculate with the time step specified in Simulink, specify a positive value for this setting. The value that is specified is used in the JMAG-RT model.	Electric signal input, voltage signal input, current signal input

Parameter	Description	Signal type
Offset Angle (deg)	Specifies the initial position of a rotation angle. The initial position of a rotation angle is specified at the position that is coincident with the d-axis direction and the direction of U-phase magnetic flux of coil as default. An offset angle can be set on the default position by specifying this parameter. The initial position of an ideal motor model (Permanent Magnet Synchronous Machine block) that is included in SimPowerSystems of MATLAB /Simulink is the electric angle 90 degrees late from the initial position of a JMAG-RT model. Therefore, specify the mechanical angle equivalent to electric angle -90 degrees (example: "-45" for four poles) as Offset Angel, when replacing a JMAG-RT model with a circuit diagram of Simulink from an ideal motor model. With this setting, the initial position of a JMAG-RT model is the electric angle 90 degrees late and a simulation can be run based on an ideal motor model and a JMAG-RT model with the same initial position.	Electric signal input, voltage signal input, current signal input



Initial position of rotation angle (default status)

Parameter	Description	Signal type
Connection type	Specifies the connection pattern. • 0: Calculates using a Y-connection • 1: Calculates using a Delta-connection	Electric signal input
Motion type	• 0: Calculates the position of the rotor by solving the motion equation based on the inertia moment and friction coefficient. • 1: Specifies the rotation of the rotor as a constant speed. The rotation speed can be specified in [Rotation Speed].	Electric signal input, voltage signal input, current signal input
Rotation speed (rpm)	Specifies the rotation speed if [Motion Type] is set to '1'.	
SetCoefficientsManually	Specifies the correction coefficient to use. Select [Set Coefficients Manually] to display the settings for the correction coefficient.	
coef_inductance	Modifies the inductance data. For example, setting coef_inductance to '2.0' doubles all the inductance data.	
coef_flux	Modifies the flux linkage of the magnet. For example, setting this to '2.0' doubles all the flux linkage data.	
Use torque coefficient file	Specified to use the current dependence correction point sequence of torque.	
coef_torque	Modifies the torque data. For example, setting this to '2.0' doubles all the torque data.	
coef_magnet	Modifies the grade of a magnet. This correction coefficient easily simulates the effects of magnets that have been changed. This coefficient affects the flux linkage of the magnets.	
coef_material	Modifies the grade of a steel sheet. This correction coefficient easily simulates the effects of materials that have been changed. This coefficient affects the inductance, flux linkage of magnets, and torque.	

Parameter	Description	Signal type
turns_coil1	Modifies the number of coil turns 1. This correction coefficient easily simulates the effects of changing the number of coil turns by specifying the number of turns. This coefficient affects the inductance, flux linkage of magnets, and torque. The number of turns for the original motor should be specified if turns_coil1 is not used.	Electric signal input, voltage signal input, current signal input
turns_coil2	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet, the current when referring to the inductance, and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0. The number of turns for the original motor should be specified if turns_coil2 is not used.	
Coil temperature	Reference temperature of coil (K)	
Magnet temperature	Reference temperature of magnet (K)	
coef_coil temperature	Temperature correction coefficient for coil (ppm/K) Range: 0.0 and over Default: 3810 ppm/K	
coef_magnet temperature	Temperature correction coefficient for magnet (%/K) Range: -100 - 0.0 (negative) Default: -0.11 %/K	

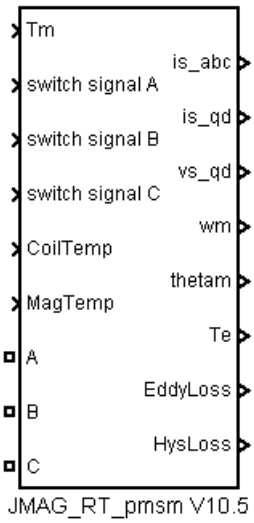
■ Input/Output of JMAG-RT model block

The input/output of a JMAG-RT model block for a three-phase PM synchronous motor is indicated in the table below.

Important **In the case that a temperature terminal of the S-Function block has no connection, the reference temperature is treated as zero for the calculation. To omit the temperature correction, connect a constant value block which is set the base temperature to the temperature terminal of the S-Function block.**

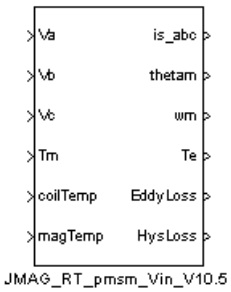
■ Electric signal input

Block	Input	Output	Input/Output
JMAG_RT_pmsm	• Tm: Load torque (N*m) • switch signal A: Switch signal of A-phase (On: 1, Off: 0) • switch signal B: Switch signal of B-phase (On: 1, Off: 0) • switch signal C: Switch signal of C-phase (On: 1, Off: 0) • coilTemp: Coil temperature (K) • magTemp: Magnet temperature (K)	• is_abc: Three-phase current (ia, ib, ic) (A) • is_qd: dq-axis current (is_q, is_d) (A) • vs_qd: dq-axis voltage (is_q, is_d) (V) • wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m) • Hys: Hysteresis loss (W) • Eddy: Eddy current loss (W) • Fuvw: Magnetic flux of magnet in no-load (Wb) • LdLq: d/q-axis inductance (H)	• A: A-phase terminal • B: B-phase terminal • C: C-phase terminal



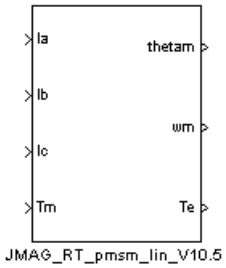
■ Voltage signal input

Block	Input	Output	Input/Output
JMAG_RT_pmsm_Vin	• Tm: Load torque (N*m) • Va: Electric potential of U-phase terminal (V) • Vb: Electric potential of V-phase terminal (V) • Vc: Electric potential of W-phase terminal (V) • coilTemp: Coil temperature (K) • magTemp: Magnet temperature (K)	• is_abc: Coil current (ia, ib, ic) (A) • thetam: Rotation angle (rad) • wm: Rotation speed (rad/s) • Te: Output torque (N*m) • Hys: Hysteresis loss (W) • Eddy: Eddy current loss (W) • Fuvw: Magnetic flux of magnet in no-load (Wb) • LdLq: d/q-axis inductance (H)	-

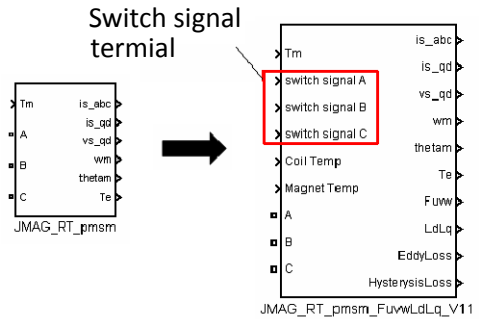


■ Current signal input

Block	Input	Output	Input/Output
JMAG_RT_pmsm_Iin	• Tm: Load torque (N*m) • Ia: Current of U-phase coils (A) • Ib: Current of V-phase coils (A) • Ic: Current of W-phase coils (A) • coilTemp: Coil temperature (K) • magTemp: Magnet temperature (K)	• thetam: Rotation angle (rad) • wm: Rotation speed (rad/s) • Te: Output torque (N*m) • Hys: Hysteresis loss (W) • Eddy: Eddy current loss (W) • Fuvw: Magnetic flux of magnet in no-load (Wb) • LdLq: d/q-axis inductance (H)	-



- Three terminals have been added to the input switch signal from JMAG-Studio Version 8.3.
  The ON/OFF signal of a switch that is connected to a, b, and c terminals can be specified for these switch signal terminals.
  - If the switch is ON, enter '1'.
  - If the switch is OFF or the terminals are open, enter '0'.

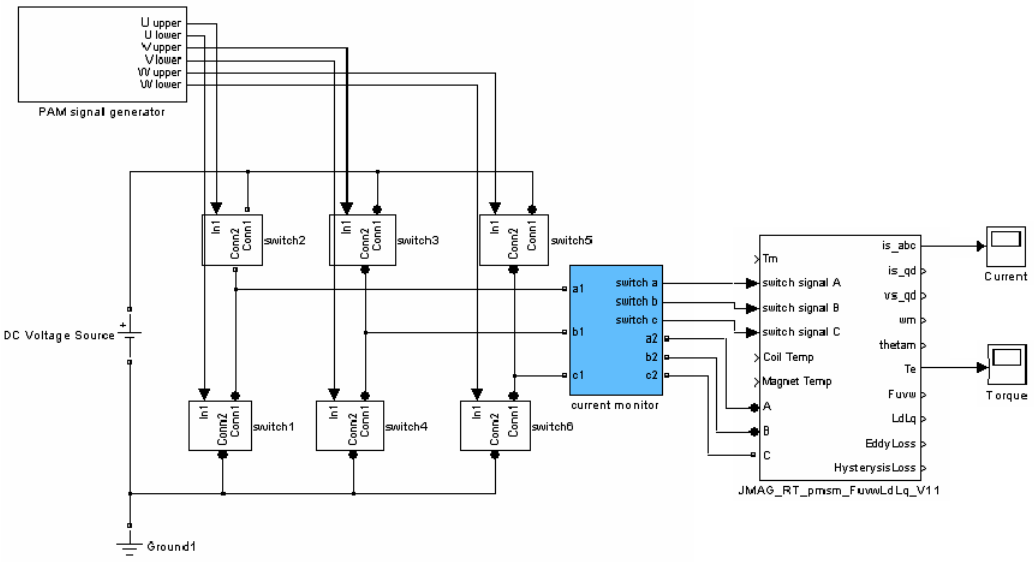


Version 8.2 or earlier Version 8.3 or later

PM Synchronous Motor Block with Switch Signal Terminal

■ Connection example 1: A JMAG-RT model block is connected to the power source via the switches

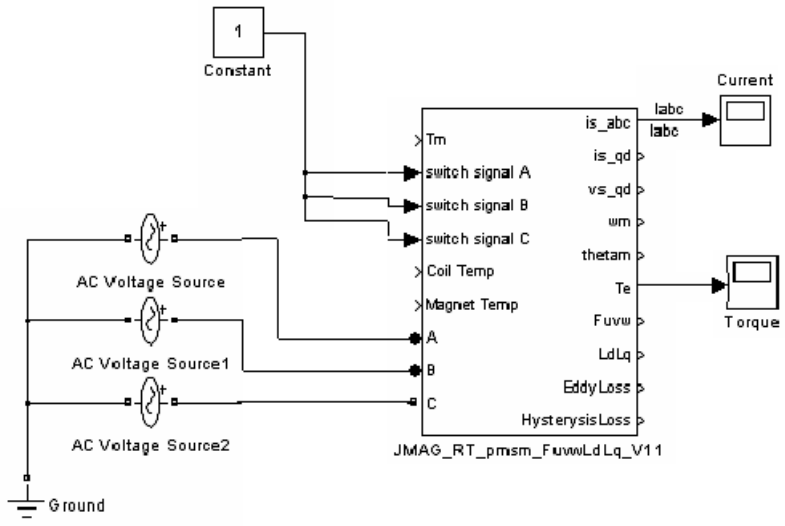
The figure below is an example of a connection where terminals a, b, and c are connected to the power source through the switches. '0' is set to the switch signal terminal if both of the switches connected to each phase are OFF. '1' is specified if either of the switches is ON. For this example, the current flowing in each phase is monitored and the switch signal terminal is '0' if the threshold is below 1.0 x 10-4A.



PAM drive

■ Connection example 2: A JMAG-RT model block is connected to the power source directly.

The terminals a, b, and c are connected to the power source without passing through the switches. Arrange the AC voltage source block, and then specify '1' to the switch signal terminal.

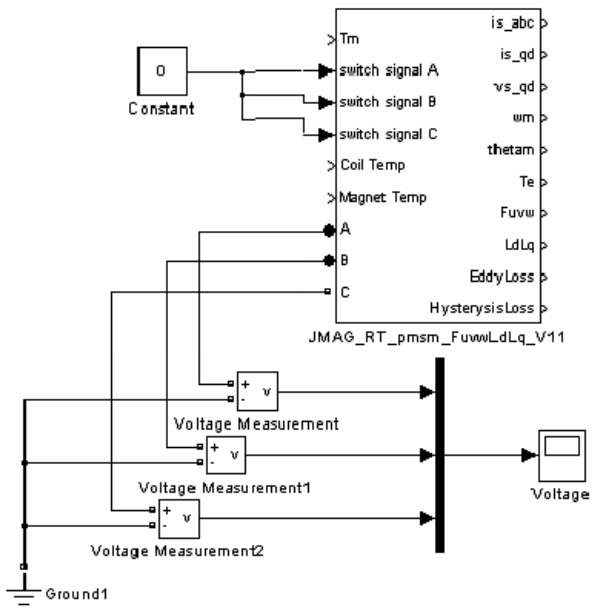


Connection without the switches

■ Connection example 3: The terminals a, b, and c of a JMAG-RT model block are open.

If the terminals a, b, and c are open, specify '0' to the switch signal terminal.

For example, specify '0' to the switch signal terminal to calculate the induced voltages as indicated in the figure below.



Induced voltage calculation

13-8-4. Circuit/Control Simulation with PSIM

Important **MagCoupler-RT, an option in PSIM, is required to use a JMAG-RT model in PSIM.**

■ Related Files

The files required for a simulation with PSIM are listed in the following table.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a model.
RT_DLL.dll	Used by the MagCoupler-RT option of PSIM
Netlist XML file (*.xml)	Defines the interfaces between the JMAG-RT model of a three-phase PM synchronous motor and the MagCoupler-RT PMSM Machine block.
Torque correction point sequence file (*.pa)	Copied when the correction point sequence of the current dependent torque is specified in the [RT Correction] tab of the JMAG-RT Library Manager. The same file name as the JMAG-RT model data file (*.rtt) is used. * This file name cannot be changed because it is used by the JMAG-RT model data file.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box in the JMAG-RT Library Manager.
- These files must be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit/control simulation. All of the files need to be updated if a model exported from an older version of JMAG-RT is re-calculated with newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation of JMAG-RT model block

The following blocks need to be used for the three-phase PM synchronous motor depending on the version of PSIM that is used.

- For an earlier version than 9.0.1, use the MagCoupler-RT PMSM Machine block. The MagCoupler-RT PMSM Machine block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [MagCoupler-RT PMSM Machine] from the menu bar.
- For version 9.0.1 and a later version, use the PM Synchronous Machine (RT) block. The PM Synchronous Machine (RT) block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [PM Synchronous Machine (RT)] from the menu bar.

A Netlist XML file needs to be specified to reference each block. PSIM displays the terminals defined by the Netlist XML file and values set for the JMAG-RT model as the default values.

■ Properties of JMAG-RT model block

The following settings can be specified for the properties of the MagCoupler-RT PMSM Machine block in PSIM.

MEMO Specifies a correction coefficient set in the JMAG-RT Library Manager as the default values in PSIM.

MEMO The initial position for the rotation angle of the motor is set to the position where the current phase is 0 (position the direction of magnetic flux in the U-phase coil is the same as the d-axis direction). The initial position of the rotation angle does not depend on the initial position of the FEM model.

Parameter	Description
MomentOfInertia	Specifies the inertia moment for a rotating machine J (kg*m^2). The calculation diverges instead of converges if this value is set to zero.
MechTimeConstant	Specifies the mechanical time constant (τ mech)
RU	Specifies the U-phase resistance (ohm)
RV	Specifies the V-phase resistance (ohm)
RW	Specifies the W-phase resistance (ohm)

Parameter	Description
coef_inertia1	Modifies the inertia moment data. For example, setting this to '2.0' doubles all the inertia moment data.
coef_inertia2	Modifies the inertia moment data. For example, setting this to '2.0' adds all the inertia moment data.
coef_inductance	Modifies the inductance data. For example, setting this to '2.0' doubles all the inductance data.
coef_flux	Modifies the flux linkage of the magnets. For example, setting this to '2.0' doubles all the flux linkage data.
coef_torque	Modifies the torque data. For example, setting this to '2.0' doubles all the torque data.
coef_magnet	Modifies the grade of a magnet. This correction coefficient easily simulates the effects of magnets that have been changed. This coefficient affects the flux linkage of the magnets.
coef_material	Modifies the grade of a steel sheet. This correction coefficient easily simulates the effects of materials that have been changed. This coefficient affects the inductance, flux linkage of magnets, and torque.
turns_coil1	Modifies the number of coil turns 1. This correction coefficient easily simulates the effects of changing the number of coil turns by specifying the number of turns. This coefficient affects the inductance, flux linkage of magnets, and torque. The number of turns for the original motor should be specified if turns_coil1 is not used.
turns_coil2	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet, the current when referring to the inductance, and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0. The number of turns for the original motor should be specified if turns_coil2 is not used.

Parameter	Description
coeftable_torque_flag	Specifies the current dependent correction point sequence of torque. Setting coeftable_torque_flag to '1' uses the correction point sequence.
Before PSIM 9.0.1: BEMF_DisplayFlag After PSIM 9.0.1: Back emf Flag	Flag for displaying back electromotive force. 1: Display the result in a graph. 0: Do not output the result.
Before PSIM 9.0.1: Angle_DisplayFlag After PSIM 9.0.1: Rotor Angle Flag	Flag for displaying the rotation angle. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: Speed_DisplayFlag After PSIM 9.0.1: Speed Flag	Flag for displaying the rotation speed. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: Torque_DisplayFlag After PSIM 9.0.1: Torque Flag	Flag for displaying the torque. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: I_DisplayFlag After PSIM 9.0.1: Current Flag	Flag for displaying the current flowing in the coil. 1: Display the result in a graph. 0: Do not output the result
Offset angle (deg)	Specifies the initial position of a rotation angle. The initial position of a rotation angle is specified at the position that is coincident with the d-axis direction and the direction of U-phase magnetic flux of coil as default. An offset angle can be set on a default position by specifying this parameter.

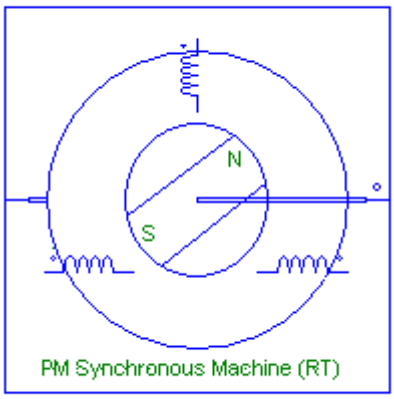
■ Input/Output of JMAG-RT model block

The input/output of the MagCoupler-RT PMSM Machine block can be set using the electric node and the rotor shaft node.

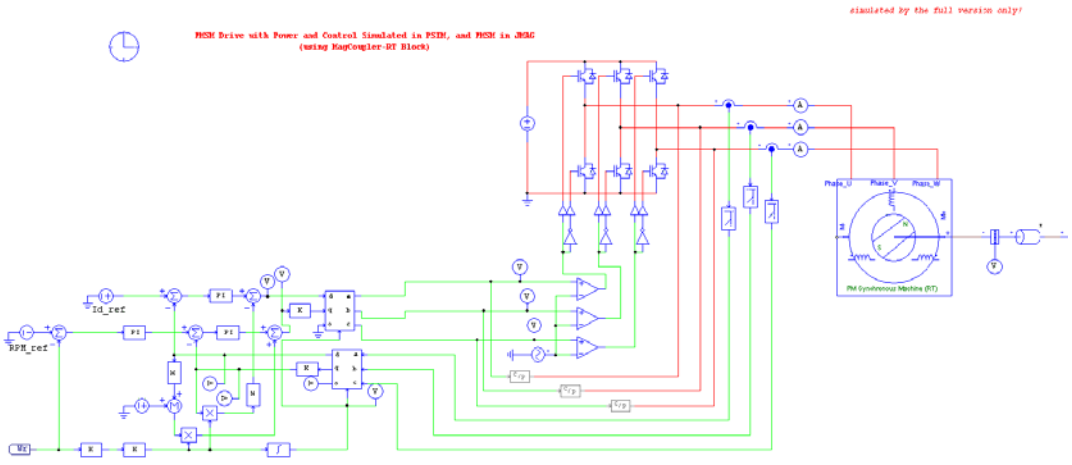
Electric node	Rotor shaft node
Connects to a phase terminal of the main circuit	Connects to a mechanical module of PSIM

Reference

For more information, see the PSIM Users Manual.



MagCoupler-RT PMSM Machine block



Example

13-8-5. Circuit/Control Simulation Using Programs in C

■ Related Files

The following files are required to run a simulation using a program in C.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor. This file is exported to the directory specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
JMAGRT_DLL.dll	Contains a dynamic link library for running a simulation using a program in C. For information about the directory where this file is stored, see P.1170 "13-15-1. Flow of Simulations Using Programs in C".

Reference

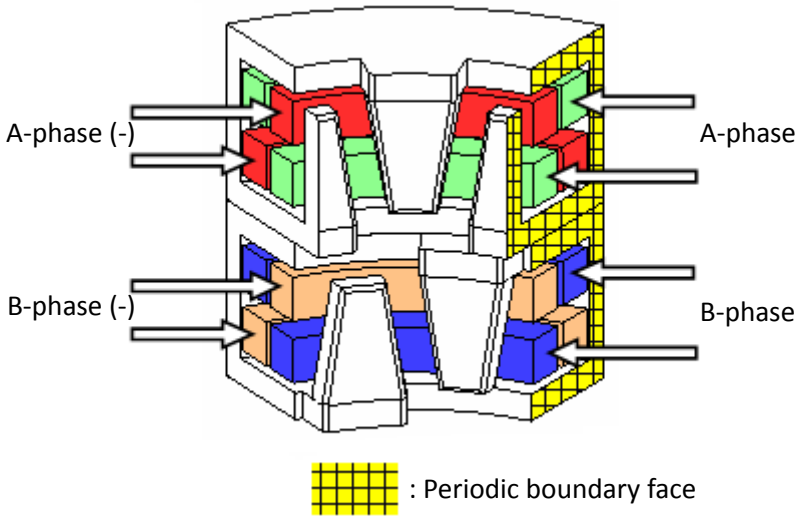
For more information about simulations using programs in C, see P.1170 "13-15. Running Simulations Using Programs in C".

13-9. Two-phase Stepping Motor

13-9-1. Creating an FEM Model with JMAG

The FEM model needs to be created satisfying the following requirements.

- Create a FEM model for a transient response analysis (TR, DP).
- Four FEM coil conditions are specified.
  Four FEM coil conditions, [A-phase], [A-phase (-)], [B-phase], and [B-phase (-)] need to be specified for a two-phase unipolar stepping motor.
  Specify the current inflow faces as indicated below for a partial model.



1/6 partial unipolar stepping motor model (60 degrees)

- The motion condition (rotation) is specified.
  - Select [Rotation] under [Type] in the [Motion] dialog box, and then select [Constant] for [Method] under [Displacement] for JMAG-Studio.
  - Select [Constant Revolution Speed] in the [Rotation Motion] settings for JMAG-Designer.
- A torque condition needs to be specified.
  - Specify an electromagnetic force and torque calculation condition, and then select the [Calculate Torque] check box when creating an FEM model in JMAG-Studio.
  - Specify a torque condition, such as nodal force, Lorentz force, or surface force, when creating an FEM model in JMAG-Designer.
- The slide condition is specified. The slide condition is automatically set when the slide mesh option is used to generate mesh.
- If a partial FEM model is used, the conversion factors for both the results and the circuit are specified.

- Specify the conversion factors in the [Analysis Control] dialog box for JMAG-Studio.
- Specify the conversion factors in the [Study Properties] dialog box for JMAG-Designer.

The results obtained from an FEM analysis are used as full model values in JMAG-RT.

The following should be noted.

- The FEM coil condition needs to be associated with an FEM coil component by creating a circuit in a standard FEM analysis, but a circuit and the FEM coil components do not need to be specified to use JMAG-RT.

13-9-2. Settings in the JMAG-RT Library Manager

This section describes how to specify the settings in the JMAG-RT Library Manager.

13-9-2-1. [Information] Tab

■ [Basic Information]

The following items can be set.

- [Title]
- [Created By]

The following items are set automatically after the model is created by JMAG-RT.

- [File Name]
- [Motor Type]
- [Creation Data]
- [JMAG Version]
- [RTT File Version]

■ [Motor Information]

The following items can be set.

- [Weight [kg]]
- [Shaft Length [m]]
- [External Diameter [m]]
- [Moment of Inertia [kgm^2]]
- [Rated Speed [rpm]]
- [Rated Power [kW]]
- [Rated Current [A]]
- [Rated Voltage [V]]

Important **These items do not affect the model creation process of JMAG-RT.**

The following items are set from the values in the [Creation] tab.

- [Number of Poles]
- [Resistance [ohm]]

Turn on the [Publish] check box to enable the display of these items in the JMAG-RT Viewer.

■ [Comment]

Any text can be entered here and will be stored with the JMAG-RT model. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Resolution]

Each item is set automatically after the model is created by JMAG-RT. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Model Folder]

The JMAG-RT model data file (*.rtt) is exported to the folder indicated in this text box. The HTML file (JCF file name.html) created when the JMAG-RT model data is created is exported to this folder with the model data.

Furthermore, a "run" folder is created in this folder when a JMAG-RT model is created. The PLOT file, etc., of the FEM analysis is exported to the run folder.

13-9-2-2. [Create] Tab

The parameters for the FEM analysis required to create a JMAG-RT model are specified in the [Create] tab.

■ [Number of Poles]

Specifies the number of poles for the motor.

■ [Configuration of FEM Coil]

- [A-Phase], [-A-Phase], [B-Phase], [-B-phase]: Displays the FEM coil conditions set for the FEM model. Specify the condition is associated with each phase.
- [Number of Turns]: Specifies the number of turns for the coil.
- [Resistance (ohm)]: Specifies the phase resistance of the coil. The [Resistance (ohm)] setting is used as the default value in the MagCoupler-RT PMSM Machine block of PSIM. This setting is not used for Simulink.

■ [Current]

Specifies the excitation and the current range.

The JMAG-RT model contains the inductance and torque data that varies with the rotation angle and current. The current waveform pattern is defined by [Current]. The time required for creating the JMAG-RT model varies with the current waveform pattern.

- [Excitation Type]: Specifies the type of excitation from the following types.
  - [One-Phase Excitation]
  - [One-Two Phase Excitation]
  - [Two-Phase Excitation]
- [Current Range (A)]: Specifies the maximum current applied to the motor. The minimum value is 0 A. The torque characteristics are obtained based on the range of minimum and maximum values.

  [Current Range (A)] is not available if the [Enable the Setting of Resolution] check box is selected.

■ [Enable the Setting of Resolution]

Reduces the scale of the model or calculation time by lowering the resolution for the linear range for a JMAG-RT model because the values within the linear range are interpolated. Whereas, you can improve the calculation accuracy by raising the resolution for the non-linear range.

Select the [Enable the Setting of Resolution] check box, and then click [Resolution] to specify the resolution. Specify the following settings in the [Resolution] dialog box.

■ [Inductance]

Calculates the current dependent inductance characteristics.

- [Number of Rotation Angle]: Specifies the number of divisions for one period of electric angle.
- [Current at Load (A)]: Specifies the range of current applied to the drive circuit with a point sequence. Click [Point Sequence] to open the [Point Sequence] dialog box, and then create a point sequence. Calculations are run as many times as the number of current values specified in [Current at Load (A)].

■ [Torque]

Specifies the rotation angle and current to obtain the torque characteristics.

- [Rotation Angle (deg)]: Specifies the rotation angle for one period of torque.
- [Division of Rotation Angle]: Specifies the number of divisions for one period of electric angle.
- [Current at Load (deg)]: Specifies the range of current applied to the motor at the time of drive with a point sequence. Click [Point sequence] to open the [Point sequence] dialog box, and then create a point sequence. Use the absolute values to define the range of current.

Click [OK] to return to the [Two-phase stepping motor] dialog box after the settings have been specified.

■ [Equation of Motion (PSIM only)]

Specifies the moment of inertia and the time constant for a JMAG-RT model if PSIM is used as a control/circuit simulator. These settings are used as the default values in the MagCoupler RT PMSM Machine block.

The JMAG-RT model created for Simulink also contains these settings, but they are not used in Simulink. These settings need to be specified again in Simulink.

A correction coefficient can be applied to the moment of inertia. The correction coefficient is defined as follows.

$$J = \mathrm{coef\_inertia\ 1} \cdot J_0 + \mathrm{coef\_inertia\ 2}$$

$$J\frac{d\omega}{dt} = T - T_{load} - B \cdot \omega$$

$$B = \frac{J}{\tau_{mech}}$$

- [Inertia Moment J_0 (kg*m^2)]: Specifies the inertia moment.
  - The value specified for [Inertia Moment for Target Region] in the [Equation of Motion] dialog box of JMAG-Studio is displayed by default.
  - The value specified for [Inertia J_1] in the [Define Equation of Motion] dialog box of JMAG-Designer is displayed by default.
- [Correction for Inertia Moment *coef_inertia 1*]: Specifies a correction coefficient for the inertia moment.
  - The value specified for [Coefficient p] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Inertia Coefficient p] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.

- [Correction for Inertia Moment *coef_inertia 2*]: Specifies a correction coefficient for inertia moment.
  - The value specified for [Correction Value J_2] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Inertia Offset J_2] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.
- [Time Constant τ_{mech} (sec)]: Specifies a time constant.

■ [Distributed Calculation]

Select the [Use Distributed Calculation] check box when distributing the calculation to multiple machines on the same network using JMAG-RT, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

Important **When using this function, it is required to set up JMAG-Remote System.**

■ [Calculation Mode]

Specifies the type of calculation to re-calculate the data that has already been calculated from the following.

- [Recalculate Only Failed Cases] : Re-calculates only the failed cases.

  The JMAG-RT model data file (*.rtt) cannot be exported until after all the cases of an analysis have been completed in JMAG-RT. The calculation time is reduced by calculating only the failed cases.

  Important **When re-calculating the failed cases, make sure to change the run folder in which the PLOT file of the FEM analysis is saved. The run folder can be created in the model folder of the [Information] tab.**

- [Recalculate with Increased Resolution]: Calculates only the cases that are added and adds the results to the existing JMAG model data file if a part of the data that has already been analyzed is modified.

  ▶▶ Reference
  For more information about adding data, see P.1028 "13-5. Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy)".

- [Redo All Calculations]: Re-calculates all of the cases even if there is only one case that fails. If there are parameters that cannot be added to a JMAG-RT model data file (*.rtt), all the cases need to be re-calculated.

■ [Do Not Output Distribution to Result File]

Specifies to output only the necessary data to the plot file when creating a JMAG-RT model ignoring the settings specified in the JCF file. If the [Do Not Output Distribution to Result File] is not selected, the output control settings specified in the JCF file are used for creating a JMAG-RT model. Note that this requires a large amount of memory.

■ [Do not leave result files]

The [Do not leave result files] is selected when running a new calculation for JMAG-RT that automatically deletes all the plot files in the working folder.

■ [Delete result file]

The plot files that are exported during calculation are deleted when a JMAG-RT model folder is already created and the [Delete result file] button is selected. The [Delete result file] button can be selected when result files that are written during the calculation exist in the run folder.

13-9-3. Circuit/Control Simulation with Simulink

■ JMAG-RT model types

There are three types of signal input in Simulink, the electric signal input, voltage signal input, and current signal input.

- A JMAG-RT model (electric signal input) can be used by connecting to the electric circuit of Simulink.
- A JMAG-RT model (voltage signal input) can be used to input voltage signal to each coil, and then output torque or current flowing in each phase without connecting to an electric circuit.
- A JMAG-RT model (current signal input) can be used to input current signal to each coil, and then output torque without connecting to a electric circuit.

MEMO The SimPowerSystems of MATLAB/Simulink is not required when a JMAG-RT model (voltage signal input or current signal input) is used.

■ Related Files

The files required for a simulation with Simulink are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor.
RT_simulink.dll	Required to use a JMAG-RT model.
JMAG_RT_2_mono_step.mdl	Defines the JMAG-RT model (electric signal input) of a stepping motor.
JMAG_RT_2_mono_step_Vin.mdl	Defines the JMAG-RT model (voltage signal input) of a stepping motor.
JMAG_RT_2_mono_step_Iin.mdl	Defines the JMAG-RT model (current signal input) of a stepping motor.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box in the JMAG-RT Library Manager.
- These files need to be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit /control simulation. All of the files listed in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

An MDL block needs to be arrange in the circuit of Simulink. An MDL block in an MDL file can be opened in the Command Window.

■ Properties of JMAG-RT model block

The following items can be specified for the properties of a JMAG-RT model block in Simulink.

Parameter	Description	Signal type
RT data file	Specifies a name for the JMAG-RT data file.	Electric input signal, voltage input signal, current input signal
Moment of inertia (Kg*m^2)	Specifies the inertia moment *J* (kg*m^2) of the rotating machine. The value specified in the JMAG-RT Library Manager is not used.	
Friction factor (Nm/(rad/s))	Specifies the friction coefficient (Nm/(rad/s)). The value specified in the JMAG-RT Library Manager is not used.	
Resistance of coil (ohm)	Specifies the resistance of the coil (ohm). The value specified in the JMAG-RT Library Manager is not used.	Electric input signal, voltage input signal
Factor	Specifies the calculation accuracy. The default value is '1000'. If the accuracy cannot be improved by reducing the time step, increase the number of digits by one. Make sure to avoid setting a value smaller than the default value. A calculation cannot achieve high accuracy if the value is too small.	Electric input signal
Sample time	Specifies the time step for the analysis. If the default value of "-1" is used, the time step specified in Simulink is used in JMAG-RT. If it takes too long to calculate with the time step specified in Simulink, specify a positive value for this parameter. The value that is specified is used in JMAG-RT.	Electric input signal, voltage input signal, current input signal

Parameter	Description	Signal type
turns_coil2 (Turnsin_coil2)	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet and the current when referring to the inductance and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0.	Electric input signal, voltage input signal, current input signal
Offset Angle (deg)	Specifies the initial position of the rotation angle. The initial position in an FEM model is used as the default position. The default position can be offset by specifying this parameter.	

■ Input/Output of JMAG-RT model block

The input/output of a JMAG-RT model block for a two-phase stepping motor.

■ Electric signal input

Block	Input	Output	Input/Output
JMAG_RT_2_mono_step	• Tm: Load torque (N*m)	• lab: Coil current (ia, -ia, ib, -ib) (A) • wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m) • Aout: Induced voltage of A-phase (V) • -Aout: Induced voltage of -A-phase (V) • Bout: Induced voltage of B-phase (V) • -Bout: Induced voltage of -B-phase (V)	• Ain: A-phase signal terminal • -Ain: -A-phase signal terminal • Bin: B-phase signal terminal • -Bin: -B-phase signal terminal

■ Voltage signal input

Block	Input	Output	Input/ Output
JMAG_RT_2_mono_step_Vin	• Tm: Load torque (N*m) • Va: Electric potential of A-phase terminal (V) • -Va: -Electric potential of -A-phase terminal (V) • Vb: Electric potential of B-phase terminal (V) • -Vb: -Electric potential of -B-phase terminal (V)	• lab: Coil current (ia, -ia, ib, -ib) (A) • wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m)	-

■ Current signal input

Block	Input	Output	Input/ Output
JMAG_RT_2_mono_step_Iin	• Tm: Load torque (N*m) • la: Current of A-phase coils (A) • -la: -Current of -A-phase coils (A) • lb: Current of B-phase coils(A) • -lb: -Current of -B-phase coils(A)	• wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m))	-

13-9-4. Circuit/Control Simulation with PSIM

Important **MagCoupler-RT, an option in PSIM, is needed to use a JMAG-RT model in PSIM.**

■ Related Files

Files needed for the analysis with PSIM are listed in the following table.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor
RT_DLL.dll	Used in MagCoupler-RT of PSIM
Netlist XML file (*.xml)	Defines the interfaces between the JMAG-RT model of a two-phase stepping motor and MagCoupler-RT.

The following should be noted about the files above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
- These files need to be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit/control simulation. All of the files in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

The following blocks need to be used for the two-phase stepping motor depending on the version of PSIM that is used.

- For an earlier version than 9.0.1, use the MagCoupler-RT block.
  The MagCoupler-RT block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [MagCoupler-RT] from the menu bar.
- For version 9.0.1 and a later version, use the Step Machine (RT) block.
  The Step Machine (RT) block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [Step Machine (RT)] from the menu bar.

A Netlist XML file needs to be specified to reference each block. PSIM displays the terminals defined by the Netlist XML file and values set for the JMAG-RT model as the default values.

■ Properties of JMAG-RT model block

The following items can be specified for the properties of the MagCoupler-RT PMSM Machine block in PSIM.

MEMO The initial position of the rotation angle of the motor is the same as the FEM model.

Parameter	Description
coef_inertia1	Modifies the inertia moment data. For example, setting this to '2.0' doubles all the inertia moment data.
coef_inertia2	Modifies the inertia moment data. For example, setting this to '2.0' adds all the inertia moment data.
MomentOfInertia	Specifies the moment of inertia for rotating machines *J* ($kg*m^2$). The calculation diverges instead of converges if MomentOfIndertia is set to '0'.
MechTimeConstant	Specifies the mechanical time constant (τ mech)
RA+	Specifies the A-phase resistance (ohm)
RA-	Specifies the -A- phase resistance (ohm)
RB+	Specifies the B-phase resistance (ohm)
RB-	Specifies the -B-phase resistance (ohm)
turns_coil2	Modifies the number of coil turns 2. The effects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet and the current when referring to the inductance and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0. The number of turns for the original motor should be specified if turns_coil2 is not used.

Parameter	Description
Before PSIM 9.0.1: BEMF_DisplayFlag After PSIM 9.0.1: Back emf Flag	Flag for displaying back electromotive force. 1: Display the result in a graph. 0: Do not output the result.
Before PSIM 9.0.1: Angle_DisplayFlag After PSIM 9.0.1: Rotor Angle Flag	Flag for displaying the rotation angle. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: Speed_DisplayFlag After PSIM 9.0.1: Speed Flag	Flag for displaying the rotation speed. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: Torque_DisplayFlag After PSIM 9.0.1: Torque Flag	Flag for displaying the torque. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: I_DisplayFlag After PSIM 9.0.1: Current Flag	Flag for displaying the current flowing in the coil. 1: Display the result in a graph. 0: Do not output the result
Offset angle (deg)	Specifies the initial position of the rotation angle. The initial position of the rotation angle is the position of the current phase at 0 by default. The offset can be specified by setting this parameter.

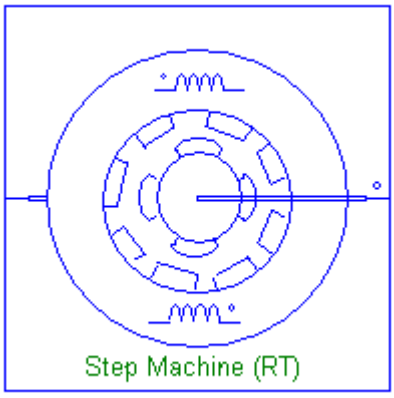
■ Input/Output of JMAG-RT model block

The input/output of the MagCoupler-RT PMSM Machine block can be set using the electric node and the rotor shaft node.

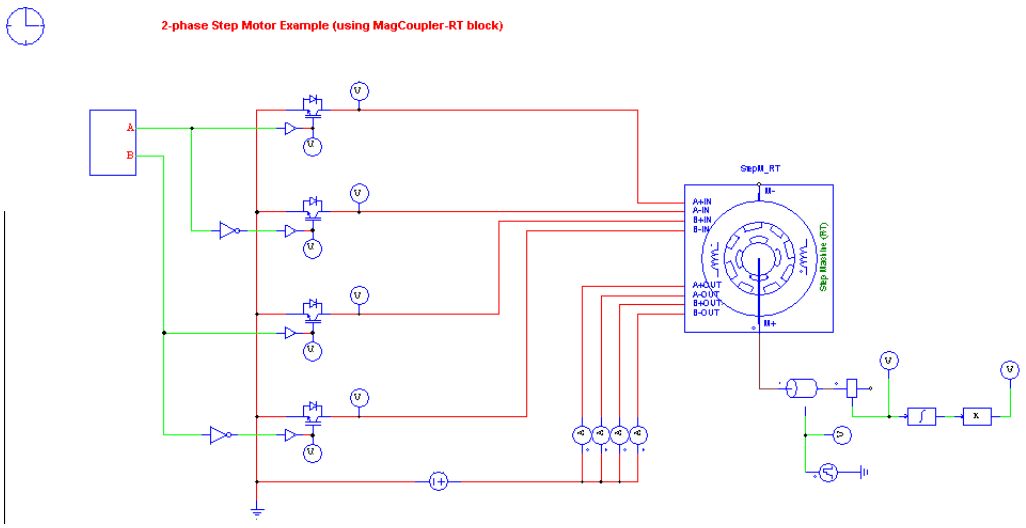
Electric node	Rotor shaft node
Connects to a phase terminal of a main circuit	Connects to a mechanical module of PSIM

Reference

For more information, see the PSIM User's Manual.



Step Machine (RT)



Example

13-9-5. Circuit/Control Simulation Using Programs in C

The following files are required to run a simulation using a program in C.

■ Related Files

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor. This file is exported to the directory specified in the [Copy to Circuit/Control Simulator] dialog box in JMAG-RT Library Manager.
JMAGRT_DLL.dll	Contains a dynamic link library that is used to run a simulation using a C language program. For information about the directory where this file is saved, seeP.1170 "13-15-1. Flow of Simulations Using Programs in C".

▶▶ Reference

For more information about simulations using a program in C, seeP.1170 "13-15. Running Simulations Using Programs in C".

13-10. Linear Solenoid

13-10-1. Creating an FEM Model with JMAG

The FEM model needs to be created satisfying the following requirements.

- Create a FEM model for a transient response analysis (TR, DP).
- One FEM coil condition is specified.
- One motion condition (translation in one direction) is specified.
- A torque condition needs to be specified.
  - Specify an electromagnetic force and torque calculation condition, and then select the [Calculate Torque] check box when creating an FEM model in JMAG-Studio.
  - Specify a torque condition, such as nodal force, Lorentz force, or surface force, when creating an FEM model in JMAG-Designer.
- The conversion factors for both the results and circuit need to be specified if a partial model is used.
  - Specify the conversion factors in the [Analysis Control] dialog box for JMAG-Studio.
  - Specify the conversion factors in the [Study Properties] dialog box for JMAG-Designer.

The results obtained from an FEM analysis are used as full model values in JMAG-RT.

The following should be noted.

- The FEM coil condition needs to be associated with an FEM coil component by creating a circuit in a standard FEM analysis, but a circuit and the FEM coil components do not need to be specified to use JMAG-RT.

13-10-2. Settings in JMAG-RT Library Manager

This section describes how to specify the settings in the JMAG-RT Library Manager.

13-10-2-1. [Information] Tab

■ [Basic Information]

The following items can be set.

- [Title]
- [Created By]

The following items are set automatically after the model is created by JMAG-RT.

- [File Name]
- [Motor Type]
- [Creation Data]
- [JMAG Version]
- [RTT File Version]

■ [Motor Information]

The following items can be set.

- [Weight [kg]]
- [Rated Speed [rpm]]
- [Rated Power [kW]]
- [Rated Current [A]]
- [Rated Voltage [V]]

Important **These items do not affect the model creation process of JMAG-RT.**

The following items are set from the values in the [Creation] tab.

- [Resistance [ohm]]

Turn on the [Publish] check box to enable the display of these items in the JMAG-RT Viewer.

■ [Comment]

Any text can be entered here and will be stored with the JMAG-RT model. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Resolution]

Each item is set automatically after the model is created by JMAG-RT. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Model Folder]

The JMAG-RT model data file (*.rtt) is exported to the folder indicated in this text box. The HTML file (JCF file name.html) created when the JMAG-RT model data is created is exported to this folder with the model data.

Furthermore, a "run" folder is created in this folder when a JMAG-RT model is created. The PLOT file, etc., of the FEM analysis is exported to the run folder.

13-10-2-2. [Create] Tab

The parameters for the FEM analysis required to create a JMAG-RT model are specified in the [Create] tab.

■ [Configuration of FEM Coil]

- [FEM Coil]: Displays the FEM coil conditions set for the FEM model.
- [Number of Turns]: Specifies the number of turns for the coil.
- [Resistance (ohm)]: Specifies the phase resistance of the coil. The [Resistance (ohm)] setting is used as the default value in the MagCoupler-RT block of PSIM. This setting is not used for Simulink.

■ [Motion Range]

Specifies the range of motion for the mover.

- [Upper Limit (m)]: Specifies the upper limit of the motion range of the mover.
- [Lower Limit (m)]: Specifies the lower limit of the motion range of the mover.

■ [Resolution (position)]

Specifies the resolution of the data for the mover. The scale of the model or calculation time can be reduced by lowering the resolution for the linear range in JMAG-RT because the values within the linear range are interpolated. The accuracy of a calculation can be improved by raising the resolution for the non-linear range.

- [Number of Divisions]: Specifies the number of divisions for the range specified in [Motion Range].
- [Use Point Sequence]: Select the [Use Point Sequence] check box, and then click [Point Sequence] to specify a point sequence. The following settings can be specified in the [Point Sequence] dialog box.
  - [Point Sequence Name]: Specifies a name for the point sequence.
  - [Point Sequence]: Specifies the position of the mover with a point sequence. Specify a value that is within the motion range.

  Click [OK] to return to the [Linear Solenoid] dialog box after the above settings are specified.

■ [Resolution (current)]

Specifies the resolution of the data for the current flowing in the linear solenoid. The scale of the model or calculation time can be reduced for the JMAG-RT model by lowering the resolution for the linear range because the values within the linear range are interpolated. The accuracy of the calculation can be improved by raising the resolution for the non-linear range.

- [Current Range (A)]: Specifies the maximum current flowing through the linear solenoid.
- [Number of Divisions]: Specifies the number of divisions for the range specified in [Current Range (A)].
- [Use Point Sequence]: Select the [Use Point Sequence] check box, and then click [Point Sequence] to specify a point sequence. The following settings can be specified in the [Point Sequence] dialog box.
  - [Point Sequence Name]: Specifies a name for the point sequence.
  - [Point Sequence]: Specifies the current with a point sequence to create the table data.

  Click [OK] to return to the [Linear Solenoid] dialog box after the above settings are specified.

■ [Equation of Motion (PSIM only)]

Specifies the mass, time constant, and spring constant for the JMAG-RT model data if PSIM is used as the control/circuit simulator. These settings are used as the default values in the MagCoupler RT block.

The JMAG-RT model created for Simulink contains these values, but they are not used in Simulink. These settings need to be specified again in Simulink.

A correction coefficient can be applied to the mass. The correction coefficient is defined as follows.

$$M = \mathrm{coef\_mass\ 1} \cdot M_0 + \mathrm{coef\_mass\ 2}$$

$$F = M\ddot{x} + B\dot{x} + Kx$$

$$B = \frac{M}{\tau_{mech}}$$

- [Mass M_0 (kg)]: Specifies the mass value.
  - The value specified for [Mass for Target Region M_1] in the [Equation of Motion] dialog box of JMAG-Studio is displayed by default.
  - The value specified for [Mass M_1] in the [Define Equation of Motion] dialog box of JMAG-Designer is displayed by default.
- [Mass Correction Coefficient *coef_mass 1*] : Specifies a correction coefficient for mass.
  - The value specified for [Coefficient p] in the [Equation of Motion] dialog box of JMAG-Studio is displayed by default.
  - The value specified for [Mass Coefficient p] in the [Define Equation of Motion] dialog box of JMAG-Designer is displayed by default.
- [Mass Correction Coefficient *coef_mass 2*]: Specifies a correction coefficient for mass.
  - The value specified for [Correction Value M_2] in the [Equation of Motion] dialog box of JMAG-Studio is displayed by default.
  - The value specified for [Mass M_2] in the [Define Equation of Motion] dialog box of JMAG-Designer is displayed by default.
- [Time Constant τ_{mech} (sec)]: Specifies a time constant.

■ [Distributed Calculation]

Select the [Use Distributed Calculation] check box when distributing the calculation to multiple machines on the same network using JMAG-RT, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

Important **When using this function, it is required to set up JMAG-Remote System.**

■ [Calculation Mode]

Specifies the type of calculation to re-calculate the data that has already been calculated from the following.

- [Recalculate Only Failed Cases] : Re-calculates only the failed cases.

  The JMAG-RT model data file (*.rtt) cannot be exported until after all the cases of an analysis have been completed in JMAG-RT. The calculation time is reduced by calculating only the failed cases.

  Important **When re-calculating the failed cases, make sure to change the run folder in which the PLOT file of the FEM analysis is saved. The run folder can be created in the model folder of the [Information] tab.**

- [Recalculate with Increased Resolution]: Calculates only the cases that are added and adds the results to the existing JMAG model data file if a part of the data that has already been analyzed is modified.

  ▶▶ Reference

  For more information about adding data, see "13-5. Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy)" on page 1028.

- [Redo All Calculations]: Re-calculates all of the cases even if there is only one case that fails. If there are parameters that cannot be added to a JMAG-RT model data file (*.rtt), all the cases need to be re-calculated.

■ [Do Not Output Distribution to Result File]

Specifies to output only the necessary data to the plot file when creating a JMAG-RT model ignoring the settings specified in the JCF file. If the [Do Not Output Distribution to Result File] is not selected, the output control settings specified in the JCF file are used for creating a JMAG-RT model. Note that this requires a large amount of memory.

■ [Do not leave result files]

The [Do not leave result files] is selected when running a new calculation for JMAG-RT that automatically deletes all the plot files in the working folder.

■ [Delete result file]

The plot files that are exported during calculation are deleted when a JMAG-RT model folder is already created and the [Delete result file] button is selected. The [Delete result file] button can be selected when result files that are written during the calculation exist in the run folder.

13-10-3. Circuit/Control Simulation with Simulink

■ Signal input types

There are three types of signal input in Simulink, the electric signal input, voltage signal input, and current signal input.

- A JMAG-RT model (electric signal input) can be used by connecting to the electric circuit of Simulink.
- A JMAG-RT model (voltage signal input) can be used to input voltage signal to each coil, and then output torque or current flowing in each phase without connecting to an electric circuit.
- A JMAG-RT model (current signal input) can be used to input current signal to each coil, and then output torque without connecting to a electric circuit.

MEMO The SimPowerSystems of MATLAB/Simulink is not required when a JMAG-RT model (voltage signal input or current signal input) is used.

■ Related Files

The files required for a simulation with Simulink are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and electromagnetic force data of a linear solenoid.
RT_simulink.dll	Required to use a JMAG-RT model.
JMAG_RT_solenoid.mdl	Defines the JMAG-RT model (electric signal input) of a linear solenoid.
JMAG_RT_solenoid_Vin.mdl	Defines the JMAG-RT model (voltage signal input) of a linear solenoid.
JMAG_RT_solenoid_Iin.mdl	Defines the JMAG-RT model (current signal input) of a linear solenoid.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box in JMAG-RT Library Manager.
- These files need to be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit /control simulation. All of the files listed in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

An MDL block needs to be arrange in the circuit of Simulink. An MDL block in an MDL file can be opened in the Command Window.

■ Properties of JMAG-RT model block

The following settings can be specified for the properties of a JMAG-RT model block in Simulink.

Parameter	Description	Signal type
RT data file	Specifies a name for the RT data file.	Electric signal input, voltage signal input, current signal input
Mass (Kg)	Specifies the mass of the mover (kg). The value specified in the JMAG-RT Library Manager is not used.	
Friction factor (N/(m/s))	Specifies the friction coefficient (N/(m/s)). The value specified in the JMAG-RT Library Manager is not used.	
Spring constant (N/m)	Specifies the spring constant (kg). The value specified in the JMAG-RT Library Manager is not used.	
Resistance of coil (ohm)	Specifies the resistance of the coil (ohm). The value specified in the JMAG-RT Library Manager is not used.	Electric signal input, voltage signal input
Factor	Specifies the calculation accuracy. The default value is '1000'. If the accuracy cannot be improved by reducing the time step, increase the number of digits by one. Make sure to avoid specifying a value smaller than the default value. The calculation cannot achieve high accuracy if the factor that is specified is too small.	Electric signal input,
Sample Time	Specifies the time step for the analysis. If the default value of "-1" is used, the time step specified in Simulink is used for the JMAG-RT model. If the calculation takes to long with the time step specified in Simulink, specify a positive value for this parameter. The value that is specified is used for the JMAG-RT model.	Electric signal input, voltage signal input, current signal input

Parameter	Description	Signal type
turns_coil2(Turns_in_coil2)	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet and the current when referring to the inductance and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0.	Electric signal input, voltage signal input, current signal input

■ Input/Output of JMAG-RT model block

The input/output of a JMAG-RT model block for a two-phase stepping motor is indicated in the table below.

■ Electric signal input

Block	Input	Output	Input/Output
JMAG_RT_solenoid	• Fm: Load (N)	• Ia: Coil current (A) • speed: Speed (m/s) • disp: Position (m) • Fe: Electromagnetic force (N)	• Ain: Electric signal phase terminal

■ Voltage signal input

Block	Input	Output	Input/Output
JMAG_RT_solenoid_Vin	• Fm: Load (N) • Va: Voltage (V)	• Ia: Coil current (A) • speed: Speed (m/s) • disp: Position (m)	-

■ Current signal input

Block	Input	Output	Input/Output
JMAG_RT_solenoid_Iin	• Fm: Load (N) • Ia: Coil current (A)	• speed: Speed (m/s) • disp: Position (m) • Fe: Electromagnetic force (N)	-

13-10-4. Circuit/Control Simulation with PSIM

Important **MagCoupler-RT, an option in PSIM, is required to use a JMAG-RT model in PSIM.**

■ Related Files

The files required for a simulation with PSIM are listed in the following table.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and electromagnetic force data of a linear solenoid.
RT_DLL.dll	Used in MagCoupler-RT of PSIM.
Netlist XML file (*.xml)	Defines the interfaces between the JMAG-RT model of a linear solenoid and the MagCoupler-RT block.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box in the JMAG-RT Library Manager.
- These files must be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit /control simulation. All of the files listed in the table need to be updated if a model exported from an older version of JMAG-RT is recalculated with newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

The following blocks need to be used for the linear solenoid depending on the version of PSIM that is used.

- For an earlier version than 9.0.1, use the MagCoupler-RT block.
  The MagCoupler-RT block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [MagCoupler-RT] from the menu bar.
- For version 9.0.1 and a later version, use the Linear Solenoid (RT) block.
  The Linear Solenoid (RT) block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [Linear Solenoid (RT)] from the menu bar.

A Netlist XML file needs to be specified to reference each block. PSIM displays the terminals defined by the Netlist XML file and values set for the JMAG-RT model as the default values.

■ Properties of JMAG-RT model block

The following settings can be specified for the properties of the MagCoupler-RT block in PSIM.

MEMO The initial position of the mover is the same as the FEM model.

Parameter	Description
coef_mass1	Modifies the mass data. For example, setting this to '2.0' doubles all the mass data.
coef_mass2	Modifies the mass data. For example, the mass data is added by setting this to '2.0'.
Mass	Specifies the mass of the mover (kg)
MechTimeConstant	Specifies the mechanical time constant (τ mech)
Spring constant	Specifies the spring constant (N/m)
DispLimitMax	Specifies the upper limit of the motion range of the mover
DispLimitMin	Specifies the lower limit of the motion range of the mover
RA	Specifies the resistance of the A-phase coil (ohm).
turns_coil2	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet and the current when referring to the inductance and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0.
Before PSIM 9.0.1: BEMF_DisplayFlag After PSIM 9.0.1: Back emf Flag	Flag for displaying back electromotive force. 1: Display the result in a graph. 0: Do not output the result.
Before PSIM 9.0.1: Angle_DisplayFlag After PSIM 9.0.1: Position Flag	Flag for displaying the displacement. 1: Display the result in a graph. 0: Do not output the result.

Parameter	Description
Before PSIM 9.0.1: Speed_DisplayFlag After PSIM 9.0.1: Velocity Flag	Flag for displaying the speed. 1: Display the result in a graph. 0: Do not output the result.
Before PSIM 9.0.1: Torque_DisplayFlag After PSIM 9.0.1: Force Flag	Flag for displaying the electromagnetic force. 1: Display the result in a graph. 0: Do not output the result.
Before PSIM 9.0.1: I_DisplayFlag After PSIM 9.0.1: Current Flag	Flag for displaying the current flowing in the coil. 1: Display the result in a graph. 0: Do not output the result.

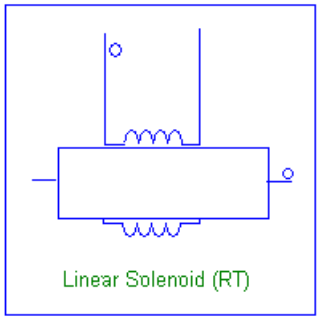
■ Input/Output of JMAG-RT model block

The input/output of the MagCoupler-RT PMSM Machine block can be set using the electric node and the rotor shaft node.

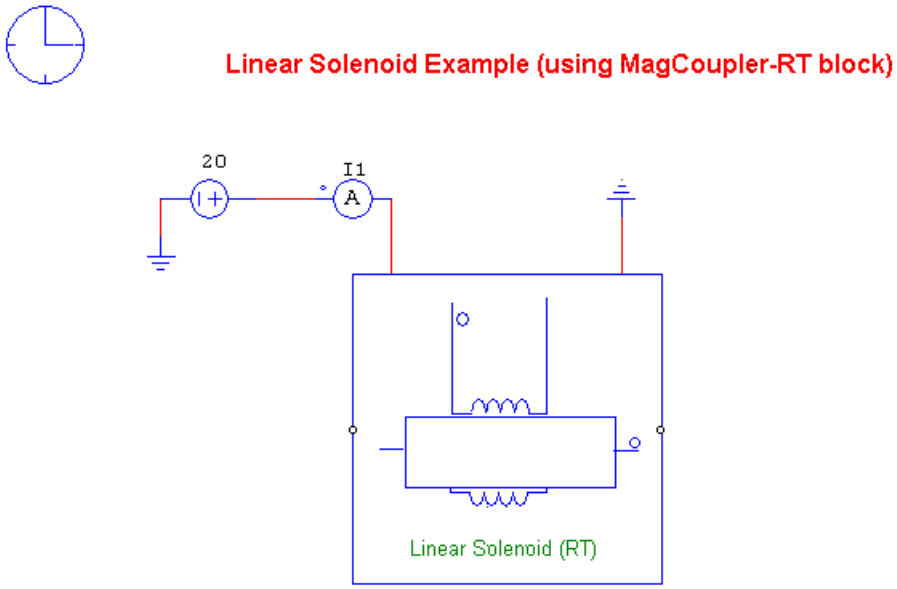
Electric node	Rotor shaft node
Connects to a phase terminal of the main circuit	Connects to a mechanical module of PSIM

▶▶ Reference

For more information, see the PSIM User's Manual.



Linear Solenoid (RT) block



Example

13-10-5. Circuit/Control Simulation Using Programs in C

■ Related Files

The following files are required to run a simulation using a program in C.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor. This file is exported to the directory specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
JMAGRT_DLL.dll	Contains a dynamic link library for running a simulation using a program in C. For information about the directory where this file is stored, see P.1170 "13-15-1. Flow of Simulations Using Programs in C".

▶▶ Reference

For more information about simulations using programs in C, see P.1170 "13-15. Running Simulations Using Programs in C".

13-11. Linear Synchronous Motor

13-11-1. Creating an FEM Model with JMAG

The FEM model needs to be created satisfying the following requirements.

- Create a FEM model for a transient response analysis (TR, DP).
- Three FEM coil conditions are specified.

  The model data cannot be imported into JMAG-RT if there are more or less than three FEM coil conditions.
- The motion condition (translation) is specified.
  - Select [Translation] under [Type] in the [Motion] dialog box of JMAG-Studio, and then select [Constant] for [Method] under [Displacement].
  - Select [Constant Speed] in the [Motion: Translation] settings of JMAG-Designer.
- A torque condition needs to be specified.
  - Specify an electromagnetic force and torque calculation condition, and then select the [Calculate Torque] check box when creating an FEM model in JMAG-Studio.
  - Specify a torque condition, such as nodal force, Lorentz force, or surface force, when creating an FEM model in JMAG-Designer.
- The conversion factors for both the results and circuit need to be specified if a partial model is used.
  - Specify the conversion factors in the [Analysis Control] dialog box for JMAG-Studio.
  - Specify the conversion factors in the [Study Properties] dialog box for JMAG-Designer.

  JMAG-RT treats the results obtained from an FEM analysis as those of a full model.
- The initial position of the mover is the same as that of the FEM model.

The following should be noted.

- The FEM coil condition needs to be associated with an FEM coil component by creating a circuit in a standard FEM analysis, but a circuit and the FEM coil components do not need to be specified to use JMAG-RT.

13-11-2. Settings in the JMAG-RT Library Manager

This section describes how to specify the settings in the JMAG-RT Library Manager.

13-11-2-1. [Information] Tab

■ [Basic Information]

The following items can be set.

- [Title]
- [Created By]

The following items are set automatically after the model is created by JMAG-RT.

- [File Name]
- [Motor Type]
- [Creation Data]
- [JMAG Version]
- [RTT File Version]

■ [Motor Information]

The following items can be set.

- [Weight [kg]]
- [Rated Speed [rpm]]
- [Rated Power [kW]]
- [Rated Current [A]]
- [Rated Voltage [V]]

Important **These items do not affect the model creation process of JMAG-RT.**

The following items are set from the values in the [Creation] tab.

- [Number of Poles]
- [Resistance [ohm]]

Turn on the [Publish] check box to enable the display of these items in the JMAG-RT Viewer.

■ [Comment]

Any text can be entered here and will be stored with the JMAG-RT model. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Resolution]

Each item is set automatically after the model is created by JMAG-RT. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Model Folder]

The JMAG-RT model data file (*.rtt) is exported to the folder indicated in this text box. The HTML file (JCF file name.html) created when the JMAG-RT model data is created is exported to this folder with the model data.

Furthermore, a "run" folder is created in this folder when a JMAG-RT model is created. The PLOT file, etc., of the FEM analysis is exported to the run folder.

13-11-2-2. [Create] Tab

The parameters for the FEM analysis required to create a JMAG-RT model are specified in the [Create] tab.

■ [Displacement per Period of Electrical Angle (m)]

Specifies the displacement for one period of electrical angle.

■ [Configuration of FEM Coil]

- [U-Phase], [V-Phase], [W-Phase]: Displays the FEM coil conditions set for the FEM model. Specify the condition associated with each phase. The motor moves in the positive direction and the phase difference between the three phases is defined relative to the U-phase. The V-phase is displaced by an electric angle of -120 degrees and the W-phase by +120 degrees.
- [Number of Turns]: Specifies the number of turns for the coil.
- [Connection Pattern]: Specifies a [Y-connection] or [Delta-connection].
- [Resistance (Ohm)]: Specifies the phase resistance of the coil. The [Resistance (ohm)] setting is used as the default value in the MagCoupler-RT block of PSIM. This setting is not used for Simulink.

■ [Drive Type]

Specifies the current waveform for the JMAG-RT model. The JMAG-RT model contains the displacement-dependent and current dependent inductance and the electromagnetic force data. The time required for creating the JMAG-RT model data varies depending on the current waveform pattern.

Select the type of current waveform of the motor to drive in the circuit/control simulator according to whether the motor has a sinusoidal or non-sinusoidal current waveform.

- [Sine Wave Drive]: Specifies a sinusoidal waveform for the current applied to the motor. Specify the following settings.
  - [Output Attraction Force]: Exports the force in the specified direction as an attraction force by selecting [X], [Y], or [Z] under [Attraction Direction].
  - [Current Amplitude Range (A)]: Specifies the current amplitude range used to create the JMAG-RT model. Specify the maximum value of the range in the [Current Amplitude Range (A)] box. The minimum value is set to 0 A.
  - [Current Phase Range (deg)]: Specifies the current phase range used to create the JMAG-RT model. Specify the maximum value of the range in the [Current Phase Range (deg)] box. The minimum value is set to 0 degrees.

    MEMO If the current exceeds the specified range when using the JMAG-RT model in a control/circuit simulator, the inductance and electromagnetic force obtained at the maximum or minimum current is applied.

  If the [Enable the Setting of Resolution] check box is selected, these settings are ignored.
- [Other]: Specifies a non-sinusoidal waveform pattern for the current applied to the motor.

  The current values of all the possible combinations specified for each displacement are applied to each coil. The JMAG-RT model data file contains the data for these current values.

  Specify the following settings if [Other] is selected.
  - [Output Attraction Force]: Exports the force in the specified direction as an attraction force by selecting [X], [Y], or [Z] under [Attraction Direction].
  - [Current Range at Load (Absolute Value) (A)]: Specifies the range of the current to be applied. Specify the maximum value of the range in the [Current Range at Load (Absolute Value) (A)] box. The minimum value is set to 0 A. The default value is 10 A.

  If the [Enable the Setting of Resolution] check box is selected, these settings are ignored.

■ [Enable the Setting of Resolution]

Reduces the scale of the model or calculation time by lowering the resolution for the linear range of a JMAG-RT model because the values within the linear range are interpolated. The calculation accuracy can be improved by raising the resolution for the non-linear range.
Select the [Enable the Setting of Resolution] check box, and then click [Resolution] to specify the resolution. Specify the following settings in the [Resolution] dialog box.

- [Inductance]: Calculates the current dependent inductance characteristics.
  - [Number of Displacement Divisions]: Specifies the number of divisions for one period of electric angle to calculate the inductance.
  - [Current at Load (deg)]: Specifies the range of current applied to the drive circuit with a point sequence. Click [Point Sequence] to open the [Point Sequence] dialog box, and then create a point sequence. Calculations are run as many times as the number of current values specified in [Current at Load (deg)].
- [Electromagnetic Force]: Specifies the displacement and current to obtain the electromagnetic force characteristic of the motor.
  - [Displacement (m)]: Specifies the displacement for one period of electromagnetic force if [Sine Wave Drive] is selected for [Drive Type]. If [Other] is selected for [Drive Type], specify the displacement for one period of electric angle.
  - [Number of Displacement Divisions]: Calculates the electromagnetic force within the specified range of displacement. Specify the number of divisions for one period of displacement.
  - [Current at Load (A)]: Specifies the range of current for the motor with a point sequence. Click [Point Sequence] to open the [Point Sequence] dialog box, and then create a point sequence. Use the absolute values to define the range of current.

    If [Sine Wave Drive] is selected for [Drive Type], specify the current amplitude. If [Other] is selected for [Drive Type], specify the possible current values including zero and the positive and negative values.
  - [Current Phase at Load (deg)]: Click [Point Sequence] to open the [Point Sequence] dialog box, and then specify the number of divisions for the current phase with a point sequence. Make sure to create a point sequence for an electric angle of 360 degrees.

    This setting is not available if [Other] is selected for [Drive Type].

Click [OK] to return to the [Linear Synchronous Motor] dialog box after the settings have been specified.

■ [Equation of Motion (PSIM only)]

Specifies the moment of inertia and the time constant for a JMAG-RT model if PSIM is used as a control/circuit simulator. These values are used as the default values in the MagCoupler RT PMSM Machine block.

The JMAG-RT model created for Simulink also contains these settings, but they are not used in Simulink. These parameters need to be specified again in Simulink.

A correction coefficient can be applied to the mass. The correction coefficient is defined as follows.

$$M = \mathrm{coef\_mass\ 1} \cdot M_0 + \mathrm{coef\_mass\ 2}$$

$$M\frac{d^2x}{dt^2} = F - F_{load} - B \cdot \frac{dx}{dt}$$

$$B = \frac{M}{\tau_{mech}}$$

- [Mass M_0 (kg)]: Specifies the mass.
  - The value specified for [Mass for Target Region M_1] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Mass M_1] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.
- [Mass Correction Coefficient *coef_mass 1*] : Specifies a correction coefficient for mass.

  The value specified for [Coefficient p] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.

  The value specified for [Mass Coefficient p] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.
- [Mass Correction Coefficient *coef_mass 2*]: Specifies a correction coefficient for mass.
  - The value specified for [Correction Value M_2] in the [Equation of Motion] dialog box for JMAG-Studio is displayed by default.
  - The value specified for [Mass M_2] in the [Define Equation of Motion] dialog box for JMAG-Designer is displayed by default.
- [Time Constant τ_{mech} (sec)]: Specifies a time constant.

■ [Distributed Calculation]

Select the [Use Distributed Calculation] check box when distributing the calculation to multiple machines on the same network using JMAG-RT, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

Important **When using this function, it is required to set up JMAG-Remote System.**

■ [Calculation Mode]

Specifies the type of calculation to re-calculate the data that has already been calculated from the following.

- [Recalculate Only Failed Cases] : Re-calculates only the failed cases.
  The JMAG-RT model data file (*.rtt) cannot be exported until after all the cases of an analysis have been completed in JMAG-RT. The calculation time is reduced by calculating only the failed cases.

  Important **When re-calculating the failed cases, make sure to change the run folder in which the PLOT file of the FEM analysis is saved. The run folder can be created in the model folder of the [Information] tab.**
- [Recalculate with Increased Resolution]: Calculates only the cases that are added and adds the results to the existing JMAG model data file if a part of the data that has already been analyzed is modified.

  ▶▶ Reference
  For more information about adding data, see P.1028 "13-5. Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy)".
- [Redo All Calculations]: Re-calculates all of the cases even if there is only one case that fails. If there are parameters that cannot be added to a JMAG-RT model data file (*.rtt), all the cases need to be re-calculated.

[Do Not Output Distribution to Result File]

Specifies to output only the necessary data to the plot file when creating a JMAG-RT model ignoring the settings specified in the JCF file. If the [Do Not Output Distribution to Result File] is not selected, the output control settings specified in the JCF file are used for creating a JMAG-RT model. Note that this requires a large amount of memory.

[Do not leave result files]

The [Do not leave result files] is selected when running a new calculation for JMAG-RT that automatically deletes all the plot files in the working folder.

[Delete result file]

The plot files that are exported during calculation are deleted when a JMAG-RT model folder is already created and the [Delete result file] button is selected. The [Delete result file] button can be selected when result files that are written during the calculation exist in the run folder.

13-11-2-3. [RT Correction] Tab

The [RT Correction] tab is displayed after a JMAG-RT model is created (after FEM analysis using JMAG-RT). The correction coefficient can be specified for each parameter exported to the JMAG-RT model data file.

MEMO The correction coefficients can also be specified in a control/circuit simulator.

MEMO The specified correction coefficients are cleared and default values are used to re-create the JMAG-RT model.

■ [Direct Correction], [Complex Correction], [Reference Correction]

The correction equation is as follows.

Correction equation for inductance: $L_{new} = K_1 \times K_5 \times \frac{K_6}{n} \times L(I_{new})$

Correction equation for linkage flux: $\Psi_{new} = K_2 \times K_4 \times K_5 \times \frac{K_6}{n} \times \frac{K_7}{n} \times \Psi$

Correction equation for electromagnetic force:

$$F_{new} = K_3 \times K_4 \times K_5^2 \times \frac{K_6}{n} \times F(I_{new})$$

Correction equation for current: $I_{new} = \frac{K_7}{n} \times I$

L_{new} : Inductance after correction (H)

Ψ_{new} : Linkage flux after correction (Wb)

F_{new}: Electromagnetic force after correction (N)

I_{new} : Reference current value after correction (A)

* Used to refer to the inductance and electromagnetic force.

L : Inductance before correction (H)

Ψ : Linkage flux before correction (Wb)

F: Electromagnetic force before correction (N)

I: Reference current value before correction (A)

n: Number of turns (turn)

Specify the following correction coefficients based on the above equations.

■ [Direct Correction]

- Inductance (coef_inductance) K_1
- Flux Linkage in Magnet (coef_flux) K_2
- Electromagnetic Force (coef_force) K_3•
- Current Dependent Electromagnetic Force Table (coeftable_force) K_{pa3}

  Current dependent point sequence is used to correct values according to the current amplitude. Select the [Use Current-Dependent Table for Electromagnetic Force Correction (coeftable_torque_flag)] check box, click [Point Sequence], and then specify the point sequence.
  The following should be noted when the current dependent point sequence is used.
  - If the current exceeds the specified range when using the JMAG-RT model in a control/circuit simulator, the inductance and torque values obtained at the maximum and minimum currents are applied instead of extrapolating.
  - The point sequence can be created by reading the data from the point sequence file (*.pa). Click [Read] in the [Point Sequence] dialog box, and then specify the point sequence file.

    ▶▶ Reference

    For more information about the point sequence file format, see P.1038 "13-8-2. Settings in the JMAG-RT Library Manager".
  - The correction point sequence data is exported to a point sequence file (*.pa) in addition to the JMAG-RT model data file (*.rtt) when the JMAG-RT model is copied to a control/circuit simulator. The same file name as the JMAG-RT model data file is used for this point sequence file.
    The name of the point sequence file is used in the JMAG-RT model data file and cannot be changed. Furthermore, the JMAG-RT model data file and point sequence file need to be stored in the same directory.

■ [Complex Correction]

- Magnet (coef_magnet) K_4
- Steel Sheet (coef_material) K_5
- Number of Turns (turns_coil1) K_6 (Turn)

■ [Reference Correction]

- Current Reference (turns_coil2) K_7 (Turn)

13-11-2-4. [Motor Characteristics] Tab

The [Motor Characteristics] tab is displayed after a JMAG-RT model is created (after the FEM analysis using JMAG-RT). Characteristics, such as torque and inductance, can be exported from the JMAG-RT model that is created in the CSV file format.

MEMO The correction coefficient specified in the [RT Correction] tab is not taken into account in the table data that is exported.

■ [Output Items]

Turn on the following check boxes and click the [Export] button.

- [Inductance (Lp, Lq)]: Exports the following CSV file. Select the [Inductance (lp, Lq)] check box, and then click [Export].
  - Ld_graph.csv, Lq_graph.csv: Contains the inductance for each displacement at the current phases in the d-axis and q-axis.
- [Thrust Force]: Exports the following CSV file. Select the [Thrust Force] check box, specify a value in the [Advance Angle (deg)] box, and then click [Export]. The following CSV files are exported.
  - Ave Force_graph.csv: Contains the average electromagnetic force for each current phase at each current value.
  - Force_graph.csv: Contains the displacement dependent electromagnetic force for each current value at the angle specified in the [Advance Angle (deg)] box.

    The force_graph.csv contains as many data points as the number of divisions for the electromagnetic force displacement specified to create a JMAG-RT model.

  MEMO [Advance Angle (deg)] indicates the phase of current.
- [Induced Voltage]: Exports the following CSV file is exported. Select the [Induced Voltage] check box, specify a value in the [Velocity of Moving Part (m/s)] box, and then click [Export].
  - InducedVoltage_graph.csv: Contains the induced voltage with no-load at the specified translation speed.

13-11-3. Circuit/Control Simulation with Simulink

■ JMAG-RT model types

There are three types of signal inputs in Simulink; electric signal input, voltage signal input, and current signal input.

- A JMAG-RT model (electric signal input) can be used by connecting to the electric circuit of Simulink.
- A JMAG-RT model (voltage signal input) can be used to input voltage signal to each coil, and then output torque or current flowing in each phase of coils without connecting an electric circuit.
- A JMAG-RT model (current signal input) can be used to input current signal to each coil, and then output torque without connecting an electric circuit.

MEMO The SimPowerSystems of MATLAB/Simulink is not required when a JMAG-RT model (voltage signal input or current signal input) is used.

■ Related Files

The files required for a simulation with Simulink are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and electromagnetic force of a motor.
RT_simulink.dll	Required to use a JMAG-RT model.
JMAG_RT_lsm.mdl	Specifies a linear synchronous motor for the JMAG-RT model. This file is exported when [Simulink (electric signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.
JMAG_RT_lsm_Vin.mdl	Specifies a linear synchronous motor for the JMAG-RT model. This file is exported when [Simulink (voltage signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.
JMAG_RT_lsm_Iin.mdl	Specifies a linear synchronous motor for the JMAG-RT model. This file is exported when [Simulink (current signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.

File name	Description
Force correction point sequence file (*.pa)	Copied when the correction point sequence for the current dependent electromagnetic force is specified in the [RT Correction] tab of the JMAG-RT Library Manager after a JMAG-RT model is created (after an FEM analysis using JMAG-RT). The same file name as the JMAG-RT model data file (*.rtt) is used. * This file name is used in the JMAG-RT model data file (*.rtt) and cannot be changed.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
- These files must be in the same folder when a circuit/control simulation is run.
- Only the files exported from the same version of JMAG-RT can be used for a circuit / control simulation. All of the files listed in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

A MDL block needs to be arrange in the circuit of Simulink. A MDL block in a MDL file can be opened in the Command Window.

■ Properties of JMAG-RT model block

The following settings can be specified for the properties of a JMAG-RT model block.

MEMO If a correction coefficient is specified in the JMAG-RT Library Manager, the value is used as the default value in Simulink.

Parameter	Description	Signal type
RT data file	Specifies a name for the RT data file.	Electric signal input, voltage signal input, current signal input
Mass (Kg)	Specifies the mass of the mover (kg). The value specified in the JMAG-RT Library Manager is not used.	
Friction factor (N/(m/s))	Specifies the friction coefficient (N/(m/s)). The value specified in the JMAG-RT Library Manager is not used.	
Resistance of coil (ohm)	Specifies the resistance of coil (ohm) The value specified in the JMAG-RT Library Manager is not used.	Electric signal input, voltage signal input
Factor	Specifies the calculation accuracy. The default value is '1000'. If the accuracy cannot be improved by reducing the time step, increase the number of digits by one. Make sure to avoid specifying a value smaller than the default value. The calculation cannot achieve high accuracy if the factor that is specified is too small.	Electric signal input,
Sample Time	Specifies the time step for the analysis. If the default value of "-1" is used, the time step specified in Simulink is used in JMAG-RT. If it takes too long to calculate with the time step specified in Simulink, specify a positive value for this setting. The value that is specified is used in the JMAG-RT model.	Electric signal input, voltage signal input, current signal input
Offset Displacement (m)	Specifies the initial position of the mover. The initial position specified in the FEM model is used. The displacement can be offset by setting this parameter. The current phase is specified based on the no-load induced voltage. "Position of the current phase at 0" indicates the "position when the d-axis current is 0."	Electric signal input

Parameter	Description	Signal type
Connection type	Specifies the connection pattern • 0: Calculates using a Y-connection • 1: Calculates using a Delta-connection	Electric signal input
Motion type	• 0: Calculates the position of the mover by solving the motion equation based on the mass and friction coefficient. • 1: Specifies the motion of the mover as a time constant. The translation speed can be specified in [Translation speed].	Electric signal input, voltage signal input, current signal input
Translation speed (m/s)	Specifies the translation speed if [Motion type] is set to '1'.	
SetCoefficientsManually	Specifies the correction coefficient to use. Selecting [Set Coefficients Manually] to display the setting for the correction coefficient.	
coef_inductance	Modifies the inductance data. For example, setting this to '2.0' doubles all inductance data.	
coef_flux	Modifies the flux linkage of the magnet. For example, setting this to '2.0' doubles all the flux linkage data.	
Use force coefficient file	Specified to use the current dependent correction point sequence of electromagnetic force.	
coef_force	Modifies the electromagnetic force data. For example, setting this to '2.0' doubles all electromagnetic force data.	
coef_magnet	Modifies the grade of a magnet. This correction coefficient easily simulates the effects of magnets that have been changed. This coefficient affects the flux linkage of magnets and electromagnetic force.	
coef_material	Modifies the grade of a steel sheet. This correction coefficient easily simulates the effects of materials that have been changed. This coefficient affects inductance, flux linkage of magnets, and electromagnetic force.	

Parameter	Description	Signal type
turns_coil1	Modifies the number of coil turns 1. This correction coefficient easily simulates the effects of changing the number of coil turns by specifying the number of turns. This coefficient affects the inductance, flux linkage of magnets and electromagnetic force. The number of turns for the original motor should be specified if turns_coil1 is not used.	Electric signal input, voltage signal input, current signal input
turns_coil2	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet, the current when referring to the inductance and the electromagnetic force. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items is multiplied by N/N0. The number of turns for the original motor should be specified if turns_coil2 is not used.	

■ Input/Output of JMAG-RT model block

The input/output of a JMAG-RT model block for a linear synchronous motor is indicated in the table below.

■ Electric signal input

Block	Input	Output	Input/Output
JMAG_RT_lsm	• Fm: Load (N) • switch signal A: Switch signal of A-phase (On: 1, Off: 0) • switch signal B: Switch signal of B-phase (On: 1, Off: 0) • switch signal C: Switch signal of C-phase (On: 1, Off: 0)	• is_abc: Coil current (ia, ib, ic) (A) • is_qd: dq-axis coil current (is_q, is_d) (A) • vs_qd: dq-axis voltage (is_q, is_d) (V) • speed: Speed (m/s) • Disp: Position (m) • Thrust: Thrust force (N) • Attractive Force: Attractive force (N)	• A: A-phase terminal • B: B-phase terminal • C: C-phase terminal

■ Voltage signal input

Block	Input	Output	Input/ Output
JMAG_RT_lsm_Vin	• Va: Electric potential of U-phase terminal (V) • Vb: Electric potential of V-phase terminal (V) • Vc: Electric potential of W-phase terminal (V) • Fm: Load (N)	• is_abc: Coil current (ia, ib, ic) (A) • Disp: Position (m) • speed: Speed (m/s) • Thrust: Thrust force (N) • Attractive Force: Attractive force (N)	-

■ Current signal input

Block	Input	Output	Input/ Output
JMAG_RT_lsm_Iin	• Ia: Current of U-phase coils (A) • Ib: Current of V-phase coils(A) • Ic: Current of W-phase coils(A) • Fm: Load (N)	• speed: Speed (m/s) • Disp: Position (m) • Thrust: Thrust force (N) • Attractive Force: Attractive force (N)	-

13-11-4. Circuit/Control Simulator with PSIM

Important **MagCoupler-RT, an option in PSIM, is required to use a JMAG-RT model in PSIM.**

■ Related Files

The files required for a simulation with PSIM are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and electromagnetic force data of a motor. *The name of the JMAG-RT model data file (*.rtt) is included in the Netlist XML file. Therefore, the name of this file cannot be changed.
RT_DLL.dll	Used in MagCoupler-RT of PSIM.
Netlist XML file (*.xml)	Defines the interfaces between the JMAG-RT model of a linear synchronous motor and MagCoupler-RT PMSM Machine.
Force correction point sequence file (*.pa)	Copied when the correction point sequence of current dependent electromagnetic force is specified in the JMAG-RT Library Manager. The same file name as the JMAG-RT model data file (*.rtt) is used. * The file name cannot be changed because it is used by the JMAG-RT data file.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
- These files must be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit /control simulation. All of the files need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

The following blocks need to be used for the linear synchronous motor depending on the version of PSIM that is used.

- For an earlier version than 9.0.1, use the MagCoupler-RT PMSM Machine block.

  The MagCoupler-RT PMSM Machine block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [MagCoupler-RT PMSM Machine] from the menu bar.
- For version 9.0.1 and a later version, use the Linear Synchronous Machine (RT) block.

  The Linear Synchronous Machine (RT) block can be arranged in the circuit of PSIM by selecting [Element] > [Power] > [MagCoupler-RT_Module] > [Linear Synchronous Machine (RT)] from the menu bar.

A Netlist XML file needs to be specified to reference the block. PSIM displays the terminals defined by the Netlist XML file and values set for the JMAG-RT model as the default values.

■ Properties of JMAG-RT model block

The following settings can be specified for the properties of the MagCoupler-RT PMSM Machine block in PSIM.

MEMO Specifies the correction coefficient set in the JMAG-RT Library Manager as the default values in PSIM.

Parameter	Description
Mass	Specifies the mass of the mover (kg)
MechTimeConstant	Specifies the mechanical time constant (τ mech)
RU	Specifies the U-phase resistance (ohm)
RV	Specifies the V-phase resistance (ohm)
RW	Specifies the W-phase resistance (ohm)
coef_inertia1	Modifies the inertia moment data. For example, setting this to '2.0' doubles all the inertia moment data.
coef_inertia2	Modifies the inertia moment data. For example, setting this to '2.0' adds all the inertia moment data.
coef_inductance	Modifies the inductance data. For example, setting this to '2.0' doubles all the inductance data.
coef_flux	Modifies the flux linkage of the magnet. For example, setting this to '2.0' doubles all the flux linkage data.
coef_force	Modifies the electromagnetic force data. For example, setting this to '2.0' doubles all the electromagnetic force data.

Parameter	Description
coef_magnet	Modifies the grade of a magnet. This correction coefficient easily simulates the effects of magnets that have been changed. This coefficient affects the flux linkage of the magnet and electromagnetic force.
coef_material	Modifies the grade of a steel sheet. This correction coefficient easily simulates the effects of materials that have been changed. This coefficient affects inductance, flux linkage of the magnet, and electromagnetic force.
coeftable_torque_flag	Specifies the current dependent correction point sequence of electromagnetic force. Setting this to '1' uses the correction point sequence.
turns_coil1	Modifies the number of coil turns 1. This correction coefficient easily simulates the effects of changing the number of coil turns by specifying the number of turns. This coefficient affects the inductance, flux linkage of the magnet, and electromagnetic force. The number of turns for the original motor should be specified if turns_coil1 is not used.
turns_coil2	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet, the current when referring to the inductance, and the current when referring to the electromagnetic force. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items is multiplied by N/ N0. The number of turns for the original motor should be specified if turns_coil2 is not used.
Before PSIM 9.0.1: BEMF_DisplayFlag After PSIM 9.0.1: Back emf Flag	Flag for displaying back electromotive force. 1: Display the result in a graph. 0: Do not output the result.
Before PSIM 9.0.1: Angle_DisplayFlag After PSIM 9.0.1: Position Flag	Flag for displaying the displacement. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: Speed_DisplayFlag After PSIM 9.0.1: Velocity Flag	Flag for displaying the speed. 1: Display the result in a graph. 0: Do not output the result

Parameter	Description
Before PSIM 9.0.1: Torque_DisplayFlag After PSIM 9.0.1: Force Flag	Flag for displaying the electromagnetic force. 1: Display the result in a graph. 0: Do not output the result
Before PSIM 9.0.1: I_DisplayFlag After PSIM 9.0.1: Current Flag	Flag for displaying the current flowing in the coil. 1: Display the result in a graph. 0: Do not output the result
Offset Disp (m)	Specifies the same initial position for the mover as that of the FEM model. The offset can be specified by setting this parameter. The current phase is specified based on the no-load induced voltage. "Position of the current phase at 0" indicates the "position when the d-axis current is 0."

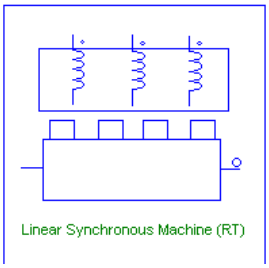
■ Input/Output of JMAG-RT model block

- The input/output of the MagCoupler-RT PMSM Machine block can be set using the electric node and the rotor shaft node.

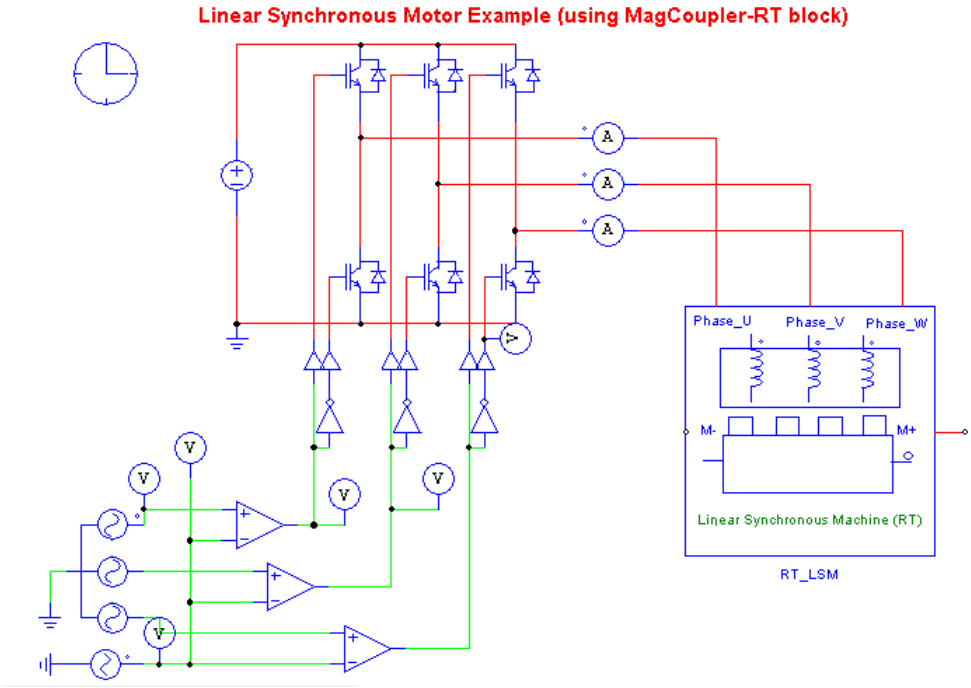
Electric node	Rotor shaft node
Connects to a phase terminal of the main circuit	Connects to a mechanical module of PSIM

▶▶ Reference

For more information, see the PSIM User's Manual.



Linear Synchronous Machine (RT) block



Example

13-11-5. Circuit/Control Simulation Using Programs in C

■ Related Files

The following files are required to run a simulation using a program in C.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and electromagnetic force data of a motor. This file is exported to the directory specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
JMAGRT_DLL.dll	Contains a dynamic link library for running a simulation using a program in C. For information about the directory where this file is stored, see P.1170 "13-15-1. Flow of Simulations Using Programs in C".

▶▶ Reference

For more information about simulations using programs in C, seeP.1170 "13-15. Running Simulations Using Programs in C".

13-12. SR Motor

13-12-1. Notes on Using the SR Motor Model Type

- The coil inductance is calculated using the current flowing only in a single phase coil. This can have an effect on the accuracy of the model in multi-phase motors where the inductance of a single phase coil is actually dependent on the current flowing in all of the phases.
- This motor type supports 3-phase to 5-phase motor.
- This motor type is not supported by PSIM.

13-12-2. Creating an FEM Model with JMAG

The FEM model needs to be created satisfying the following requirements.

- Create a FEM model for a transient response analysis (TR, DP).
- Specify the same number of FEM coil conditions in the JCF file as the number of phases of the SR motor.
- The motion condition (rotation) is specified.
  - Select [Rotation] under [Type] in the [Motion] dialog box, and then select [Constant] for [Method] under [Displacement] for JMAG-Studio.
  - Select [Constant Revolution Speed] in the [Rotation Motion] settings for JMAG-Designer.
- It is assumed that the initial position of the FEM model is zero when calculating an inductance and torque. Therefore the center of the A-phase must be put on the non-aligned position.

- A torque condition needs to be specified.
  - Specify an electromagnetic force and torque calculation condition, and then select the [Calculate Torque] check box when creating an FEM model in JMAG-Studio.
  - Specify a torque condition, such as nodal force, Lorentz force, or surface force, when creating an FEM model in JMAG-Designer.
- The slide condition is specified. The slide condition is automatically set when the slide mesh option is used to generate mesh.
- If a partial FEM model is used, the conversion factors for both the results and the circuit are specified.
  - Specify the conversion factors in the [Analysis Control] dialog box for JMAG-Studio.
  - Specify the conversion factors in the [Study Properties] dialog box for JMAG-Designer.

  The results obtained from an FEM analysis are used as full model values in JMAG-RT.

  The following should be noted.
- The FEM coil condition needs to be associated with an FEM coil component by creating a circuit in a standard FEM analysis, but a circuit and the FEM coil components do not need to be specified to use JMAG-RT.

13-12-3. Settings in the JMAG-RT Library Manager

This section describes how to specify the settings in the JMAG-RT Library Manager.

13-12-3-1. [Information] Tab

■ [Basic Information]

The following items can be set.

- [Title]
- [Created By]

The following items are set automatically after the model is created by JMAG-RT.

- [File Name]
- [Motor Type]
- [Creation Data]
- [JMAG Version]
- [RTT File Version]

■ [Motor Information]

The following items can be set.

- [Weight [kg]]
- [Shaft Length [m]]
- [External Diameter [m]]
- [Rated Speed [rpm]]
- [Rated Power [kW]]
- [Rated Current [A]]
- [Rated Voltage [V]]
- [Moment of Inertia [kgm^2]]

Important **These items do not affect the model creation process of JMAG-RT.**

The following items are set from the values in the [Creation] tab.

- [Number of Poles]
- [Resistance [ohm]]

Turn on [Publish] check box to enable the display of these items in the JMAG-RT Viewer.

■ [Comment]

Any text can be entered here and will be stored with the JMAG-RT model. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Resolution]

Each item is set automatically after the model is created by JMAG-RT. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Model Folder]

The JMAG-RT model data file (*.rtt) is exported to the folder indicated in this text box. The HTML file (JCF file name.html) created when the JMAG-RT model data is created is exported to this folder with the model data.

Furthermore, a “run” folder is created in this folder when a JMAG-RT model is created. The PLOT file, etc., of the FEM analysis is exported to the run folder.

13-12-3-2. [Create] Tab

The parameters for the FEM analysis required to create a JMAG-RT model are specified in the [Create] tab.

■ [Number of Poles]

Specifies the number of poles for the motor.

■ [Number of Phases]

Specifies the number of phases for the motor from [3 phase], [4 phase] and [5 phase].

■ [Configuration of FEM Coil]

- [A-Phase], [B-Phase], [C-Phase], [D-Phase], [E-Phase]: Displays the FEM coil conditions set for the FEM model. Specify the condition is associated with each phase.
- [Number of Turns]: Specifies the number of turns for the coil.
- [Resistance (ohm)]: Specifies the phase resistance of the coil. The [Resistance (ohm)] setting is used as the default value in the MagCoupler-RT PMSM Machine block of PSIM. This setting is not used for Simulink.

■ [Current]

- [Simple Settings]:

  This setting causes the number of points in the JMAG-RT model to be set automatically from the range of rated current.

  The following item is specified for this setting.

  - [Rated Current (A)]

  MEMO The JMAG-RT model that is created has a current range between 0 A and three times the specified rated current. Seven data points are specified for the current.

- [Advanced Settings]:

  This setting specifies the dependency of current amplitude and current phase using a point sequence.

  Click [Setting] then [Advanced Settings of Current] is displayed.

  Specify the following items for the inductance setting.

  - [Amplitude (A)]: Click [Point Sequence] then specify a point sequence of current amplitude.
  - [Mechanical Angle Resolution (deg)]

  MEMO The [Create Point Sequence] dialog box is displayed by clicking [Create Point Sequence] in the [Current Point Sequence] dialog box. Specify [Min], [Max], and [Division], and then click [OK]. The point sequence that is created is displayed in each point sequence dialog box.

  Specify the following items for the torque setting.

  - [Amplitude (A)]: Click [Point Sequence] then specify a point sequence for the current amplitude.
  - [Mechanical Angle Resolution (deg)]
  - [Calculate Rotation Angle from the Number of Poles]: If the periodicity of torque ripple can be determined from the number of poles, select the [Calculate Rotation Angle from the Number of Poles] check box. The angle per period is automatically calculated.

■ [Distributed Calculation]

Select the [Use Distributed Calculation] check box when distributing the calculation to multiple machines on the same network using JMAG-RT, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

Important **When using this function, it is required to set up JMAG-Remote System.**

■ [Calculation Mode]

Specifies the type of calculation to re-calculate the data that has already been calculated from the following.

- [Recalculate Only Failed Cases] : Re-calculates only the failed cases.

  The JMAG-RT model data file (*.rtt) cannot be exported until after all the cases of an analysis have been completed in JMAG-RT. The calculation time is reduced by calculating only the failed cases.

  Important **When re-calculating the failed cases, make sure to change the run folder in which the PLOT file of the FEM analysis is saved. The run folder can be created in the model folder of the [Information] tab.**

- [Recalculate with Increased Resolution]: Calculates only the cases that are added and adds the results to the existing JMAG model data file if a part of the data that has already been analyzed is modified.

  ▶▶ Reference

  For more information about adding data, see "13-5. Adding Data to the JMAG-RT Model (Re-calculating after Improving the Accuracy)" on page 1028.

- [Redo All Calculations]: Re-calculates all of the cases even if there is only one case that fails. If there are parameters that cannot be added to a JMAG-RT model data file (*.rtt), all the cases need to be re-calculated.

■ [Do Not Output Distribution to Result File]

Specifies to output only the necessary data to the plot file when creating a JMAG-RT model ignoring the settings specified in the JCF file. If the [Do Not Output Distribution to Result File] is not selected, the output control settings specified in the JCF file are used for creating a JMAG-RT model. Note that this requires a large amount of memory.

■ [Do not leave result files]

The [Do not leave result files] is selected when running a new calculation for JMAG-RT that automatically deletes all the plot files in the working folder.

■ [Delete result file]

The plot files that are exported during calculation are deleted when a JMAG-RT model folder is already created and the [Delete result file] button is selected. The [Delete result file] button can be selected when result files that are written during the calculation exist in the run folder.

13-12-4. Circuit/Control Simulation with Simulink

■ Signal Types

There are three types of signal input in Simulink, the electric signal input, voltage signal input, and current signal input.

- A JMAG-RT model (electric signal input) can be used by connecting to the electric circuit of Simulink.
- A JMAG-RT model (voltage signal input) can be used to input voltage signal to each coil, and then output torque or current flowing in each phase without connecting to an electric circuit.
- A JMAG-RT model (current signal input) can be used to input current signal to each coil, and then output torque without connecting to a electric circuit.

MEMO The SimPowerSystems of MATLAB/Simulink is not required when a JMAG-RT model (voltage signal input or current signal input) is used.

■ Related Files

The files required for a simulation with Simulink are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor.
RT_simulink.dll	Required to use a JMAG-RT model.
3 phase: JMAG_RT_SR_3phase.mdl 4 phase: JMAG_RT_SR_4phase.mdl 5 phase: JMAG_RT_SR_5phase.mdl	Defines the JMAG-RT model (electric signal input) of a SR motor.
3 phase: JMAG_RT_SR_3phase_Vin.mdl 4 phase: JMAG_RT_SR_4phase_Vin.mdl 5 phase: JMAG_RT_SR_5phase_Vin.mdl	Defines the JMAG-RT model (voltage signal input) of a SR motor.

File name	Description
3 phase: JMAG_RT_SR_3phase_lin.mdl 4 phase: JMAG_RT_SR_4phase_lin.mdl 5 phase: JMAG_RT_SR_5phase_lin.mdl	Defines the JMAG-RT model (current signal input) of a SR motor.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box in the JMAG-RT Library Manager.
- These files need to be in the same folder when a circuit/control simulation is run.
- Only the files that are exported from the same version of JMAG-RT can be used for a circuit /control simulation. All of the files listed in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

An MDL block needs to be arrange in the circuit of Simulink. An MDL block in an MDL file can be opened in the Command Window.

■ Properties of JMAG-RT model block

The following items can be specified for the properties of a JMAG-RT model block in Simulink.

Parameter	Description	Signal type
RT data file	Specifies a name for the JMAG-RT data file.	Electric input signal, voltage input signal, current input signal
Moment of inertia (Kg*m^2)	Specifies the inertia moment J (kg*m^2) of the rotating machine. The value specified in the JMAG-RT Library Manager is not used.	
Friction factor (Nm/(rad/s))	Specifies the friction coefficient (Nm/(rad/s)). The value specified in the JMAG-RT Library Manager is not used.	
Resistance of coil (ohm)	Specifies the resistance of the coil (ohm). The value specified in the JMAG-RT Library Manager is not used.	Electric input signal, voltage input signal

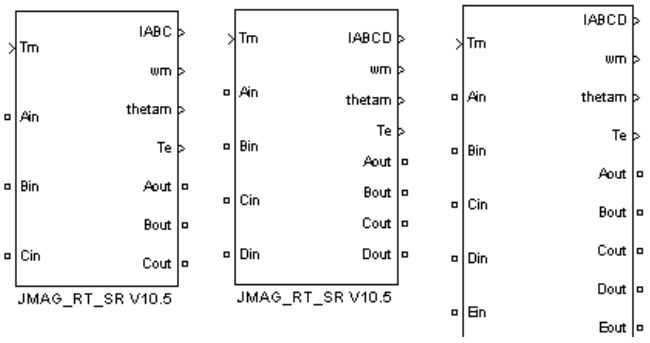
Parameter	Description	Signal type
Factor	Specifies the calculation accuracy. The default value is '1000'. If the accuracy cannot be improved by reducing the time step, increase the number of digits by one. Make sure to avoid setting a value smaller than the default value. A calculation cannot achieve high accuracy if the value is too small.	Electric input signal
Sample time	Specifies the time step for the analysis. If the default value of "-1" is used, the time step specified in Simulink is used in JMAG-RT. If it takes too long to calculate with the time step specified in Simulink, specify a positive value for this parameter. The value that is specified is used in JMAG-RT.	Electric input signal, voltage input signal, current input signal
turns_coil2 (Turnsin_coil2)	Modifies the number of coil turns 2. The affects of changing the number of coil turns can be easily simulated by specifying the number of coil turns for one coil phase. This correction coefficient affects the flux linkage of the magnet and the current when referring to the inductance and the torque. When the number of turns specified in the JMAG-RT Library Manager is expressed as N0, and turns_coil2 as N, these items are multiplied by N/ N0.	
Offset Angle (deg)	Specifies the initial position of the rotation angle. The initial position in an FEM model is used as the default position. The default position can be offset by specifying this parameter.	

■ Input/Output of JMAG-RT model block

The input/output of a JMAG-RT model block for a two-phase stepping motor.

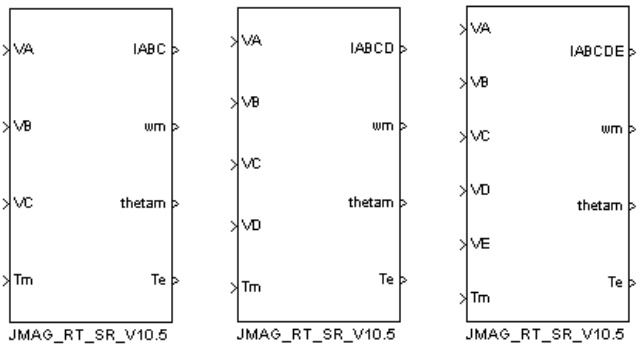
■ Electric signal input

Block	Input	Output	Input/Output
3 phase: JMAG_RT_SR_3phase 4 phase: JMAG_RT_SR_4phase 5 phase: JMAG_RT_SR_5phase	• Tm: Load torque (N*m)	• lab: Coil current (ia, -ia, ib, -ib) (A) • wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m)	• Ain: A-phase signal terminal • Bin: B-phase signal terminal • Cin: C-phase signal terminal • Din: D-phase signal terminal • Ein: E-phase signal terminal



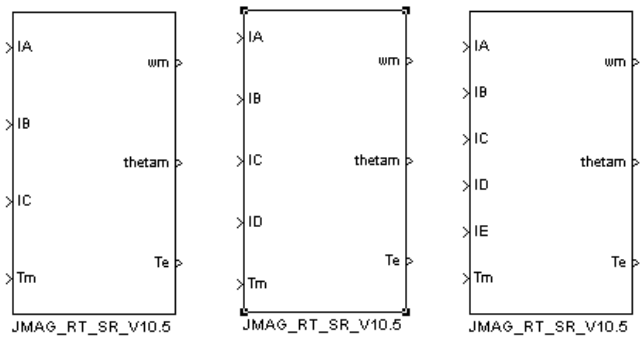
■ Voltage signal input

Block	Input	Output	Input/ Output
3 phase: JMAG_RT_SR_ 3phase_Vin 4 phase: JMAG_RT_SR_ 4phase_Vin 5 phase: JMAG_RT_SR_ 5phase_Vin	• Tm: Load torque (N*m) • Va: Electric potential of A-phase terminal (V) • Vb: Electric potential of B-phase terminal (V) • Vc: Electric potential of C-phase terminal (V) • Vd: Electric potential of D-phase terminal (V) • Ve: Electric potential of E-phase terminal (V)	• lab: Coil current (ia, -ia, ib, -ib) (A) • wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m)	-



■ Current signal input

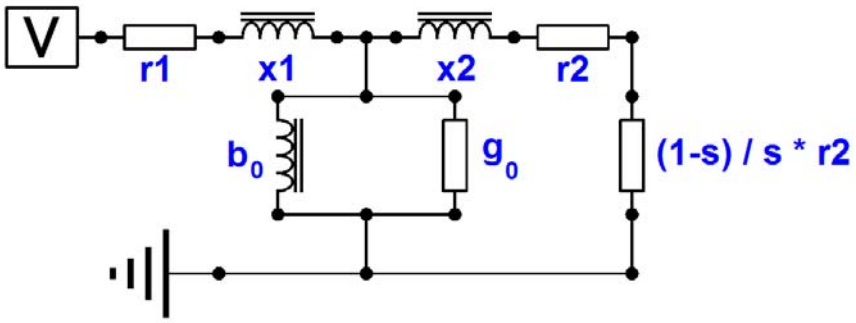
Block	Input	Output	Input/ Output
3 phase: JMAG_RT_SR_ 3phase_lin 4 phase: JMAG_RT_SR_ 4phase_lin 5 phase: JMAG_RT_SR_ 5phase_lin	• Tm: Load torque (N*m) • Ia: Current of A-phase coils (A) • Ib: Current of B-phase coils(A) • Ic: Current of C-phase coils(A) • Id: Current of D-phase coils(A) • Ie: Current of E-phase coils(A)	• wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m))	-



13-13. Induction Motor

13-13-1. Notes on Using the Induction Motor Model Type

- This motor type is not supported by PSIM
- The motor type only supports three phase induction motor.
- A model is created on a T-type equivalent circuit using an analysis value of a constraint state and a synchronized state. The circuit parameter is calculated by interpolating in the linear interpolation between a constraint state and a synchronized state. The following are the equivalent circuits that are used.



T-type equivalent circuit for an induction motor

  - x1: Primary leakage reactance (Ohm)
  - x2: Secondary leakage reactance (Ohm)
  - r1: Primary resistance (Ohm)
  - r2: Secondary resistance (Ohm)
  - g_0: Iron loss equivalent conductance (S)
  - b_0: Excitation susceptance (S)
  - s: Slip
- Higher harmonic component cannot be considered
- The accuracy is affected if a different synchronous speed is used in the control/circuit simulator from that for which the model was created using JMAG-RT.

13-13-2. Creating an FEM Model with JMAG

An FEM model needs to be created to meet the following requirements.

- Create an FEM model for a frequency response analysis (FQ).
- Three FEM coil conditions need to be specified.
  The FEM model cannot be imported into JMAG-RT if there are more or less than three FEM coil conditions.
- A rotation motion condition needs to be specified.
  - Select [Rotation] under [Type] in the [Motion] dialog box for JMAG-Studio. Furthermore, select [Constant] for [Method] under [Displacement].
  - Select [Constant Revolution Speed] in the [Rotation Motion] settings for JMAG-Designer.

  JMAG-RT uses the FEM model settings described above even if the settings that are specified differ.
- A torque condition needs to be specified.
  - Specify an electromagnetic force and torque calculation condition, and then select the [Calculate Torque] check box when creating an FEM model in JMAG-Studio.
  - Specify a torque condition, such as nodal force, Lorentz force, or surface force, when creating an FEM model in JMAG-Designer.
- Setting an Ironloss condition in the JCF file is needed when considering the ironloss effect.
- The conversion factors for both the results and the circuit need to be specified if a partial model is used.
  - Specify the conversion factors in the [Analysis Control] dialog box for JMAG-Studio.
  - Specify the conversion factors in the [Study Properties] dialog box for JMAG-Designer.

  JMAG-RT uses the results obtained from an FEM analysis as full model values.

The following should be noted.

- The FEM coil condition needs to be associated with an FEM coil component by creating a circuit in a standard FEM analysis, but a circuit and the FEM coil components do not need to be specified to use JMAG-RT.

13-13-3. Settings in the JMAG-RT Library Manager

Specify the following items.

13-13-3-1. [Information] Tab

The [Information] tab manages information in the JMAG-RT model.

■ [Basic Information]

The following items can be set.

- [Title]
- [Created By]

The following items are set automatically after the model is created by JMAG-RT.

- [File Name]
- [Motor Type]
- [Creation Data]
- [JMAG Version]
- [RTT File Version]

■ [Motor Information]

The following items can be set.

- [Weight [kg]]
- [Shaft Length [m]]
- [External Diameter [m]]
- [Moment of Inertia [kgm^2]]
- [Rated Speed [rpm]]
- [Rated Power [kW]]
- [Rated Current [A]]
- [Rated Voltage [V]]

Important **These items do not affect the model creation process of JMAG-RT.**

The following items are set from the values in the [Creation] tab.

- [Number of Poles]

Turn on [Publish] check box to enable the display of these items in the JMAG-RT Viewer.

■ [Comment]

Any text can be entered here and will be stored with the JMAG-RT model. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Resolution]

Each item is set automatically after the model is created by JMAG-RT. Turn on [Publish] check box to enable the display in the JMAG-RT Viewer.

■ [Model Folder]

The JMAG-RT model data file (*.rtt) is exported to the folder indicated in this text box. The HTML file (JCF file name.html) created when the JMAG-RT model data is created is exported to this folder with the model data.

Furthermore, a "run" folder is created in this folder when a JMAG-RT model is created. The PLOT file, etc., of the FEM analysis is exported to the run folder.

13-13-3-2. [Create] tab

The [Create] tab specifies the parameters for the FEM analysis required to create a JMAG-RT model.

■ [Number of Poles]

Specifies the number of poles for the motor.

■ [Model Divisions]

Specifies the number of divisions for the model. For example, "1" is specified in the case of a full model. "4" is specified in the case of a quarter model.

■ [Configuration of FEM Coil]

- [U-Phase], [V-Phase], [W-Phase]: Displays the FEM coil conditions set for the FEM model. Specify the condition is associated with each phase.
- [Number of Turns]: Specifies the number of turns for the coil.
- [Resistance (ohm)]: Specifies the phase resistance of the coil. [Resistance (ohm)] is not used for Simulink.

■ [Rated]

- [Frequency (Hz)]: Specifies the rated frequency.
- [Voltage (V)]: Specifies the rated voltage.

■ [Ironloss]

Select the [Allow Iron Loss] check box when using an ironloss effect.

■ [Distributed Calculation]

Select the [Use Distributed Calculation] check box when distributing the calculation to multiple machines on the same network using JMAG-RT, and then select the CPU group to use for the calculation from the [CPU Group Name] list.

Important **When using this function, it is required to set up JMAG-Remote System.**

■ [Calculation Mode]

Specifies the type of calculation to re-calculate the data that has already been calculated from the following.

- [Redo All Calculations]: Re-calculates all of the cases even if there is only one case that fails. If there are parameters that cannot be added to a JMAG-RT model data file (*.rtt), all the cases need to be re-calculated.

■ [Do Not Output Distribution to Result File]

Specifies to output only the necessary data to the plot file when creating a JMAG-RT model ignoring the settings specified in the JCF file. If the [Do Not Output Distribution to Result File] is not selected, the output control settings specified in the JCF file are used for creating a JMAG-RT model. Note that this requires a large amount of memory.

■ [Do not leave result files]

The [Do not leave result files] is selected when running a new calculation for JMAG-RT that automatically deletes all the plot files in the working folder.

■ [Delete result file]

The plot files that are exported during calculation are deleted when a JMAG-RT model folder is already created and the [Delete result file] button is selected. The [Delete result file] button can be selected when result files that are written during the calculation exist in the run folder.

13-13-4. Circuit/Control Simulation with Simulink

■ Signal Types

There are three types of signal inputs in Simulink; electric signal input, voltage signal input, and current signal input.

- A JMAG-RT model (electric signal input) can be used by connecting to the electric circuit of Simulink.
- A JMAG-RT model (voltage signal input) can be used to input the voltage signal to each coil, and then output the torque or current flowing in each phase of coils without connecting an electric circuit.
- A JMAG-RT model (current signal input) can be used to input the current signal to each coil, and then output torque without connecting an electric circuit.

MEMO The SimPowerSystems of MATLAB/Simulink is not required when a JMAG-RT model (voltage signal input or current signal input) is used.

■ Related Files

The files required for a simulation with Simulink are listed in the table below.

File name	Description
JMAG-RT model data file (*.rtt)	Contains the inductance and torque data of a motor.
RT_simulink.dll	Required to use a JMAG-RT model in Simulink.
JMAG_RT_IM.mdl	Specifies a induction motor for the JMAG-RT model (electric signal input). This file is exported when [Simulink (electric signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.
JMAG_RT_IM_Vin.mdl	Specifies a induction motor for the JMAG-RT model (voltage signal input). This file is exported when [Simulink (voltage signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.
JMAG_RT_IM_Iin.mdl	Specifies a induction motor for the JMAG-RT model (current signal input). This file is exported when [Simulink (current signal input)] is selected for [Platform] in the [Copy to Circuit/Control Simulator] dialog box.

The following should be noted when using the files described above.

- These files are exported to the folder specified in the [Copy to Circuit/Control Simulator] dialog box of the JMAG-RT Library Manager.
- These files must be in the same directory when a circuit/control simulation is run.

- Only the files exported from the same version of JMAG-RT can be used for a circuit / control simulation. All of the files listed in the table above need to be updated if a model exported from an older version of JMAG-RT is re-calculated with a newer version of JMAG-RT or a circuit/control simulation may not run properly. The JMAG-RT model block in the circuit of Simulink also needs to be updated.

■ Operation for JMAG-RT model block

A MDL block needs to be arrange in the circuit of Simulink. A MDL block in a MDL file can be opened in the Command Window.

■ Properties of JMAG-RT model block

The following settings can be specified for the properties of a JMAG-RT model block.

Parameter	Description	Signal type
RT data file	Specifies a name for the JMAG-RT data file.	Electric signal input, voltage signal input, current signal input
Use IronLoss	Flag for using iron loss Results for the iron loss can be obtained if a JMAG-RT model data file with iron loss information is used to run a calculation and the [Use IronLoss] check box is selected. Iron loss information is not taken into account when the [Use IronLoss] check box is selected and a JMAG-RT model data file without iron loss information is used.	
Moment of inertia (Kg*m^2)	Specifies the inertia moment (Kg*m^2) of the rotating machine. The value specified in the JMAG-RT Library Manager is not used.	
Friction factor (Nm/(rad/s))	Specifies the friction coefficient (Nm/(rad/s)). The value specified in JMAG-RT Library Manager is not used.	
Factor	Specifies the calculation accuracy. The default value is '1000'. If the accuracy cannot be improved by increasing the time step, increase the number of digits by one. Make sure to avoid specifying a value smaller than the default value. The calculation cannot achieve high accuracy if the factor that is specified is too small.	Electric signal input

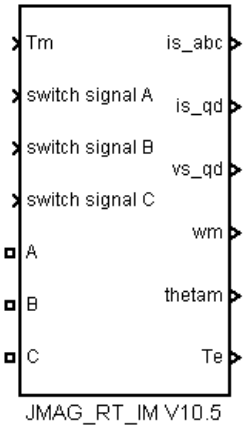
Parameter	Description	Signal type
Sample Time	Specifies the time step for the analysis. If the default value of "-1" is used, the time step specified in Simulink is used in JMAG-RT. If it takes too long to calculate with the time step specified in Simulink, specify a positive value for this setting. The value that is specified is used in the JMAG-RT model.	Electric signal input, voltage signal input, current signal input
Motion type	• 0: Calculates the position of the rotor by solving the motion equation based on the inertia moment and friction coefficient. • 1: Specifies the rotation of the rotor as a constant speed. The rotation speed can be specified in [Rotation Speed].	
Rotation speed (rpm)	Specifies the rotation speed if [Motion Type] is set to '1'.	

■ Input/Output of JMAG-RT model block

The input/output of a JMAG-RT model block for a three-phase PM synchronous motor is indicated in the table below.

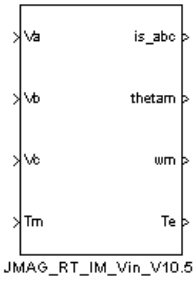
■ Electric signal input

Block	Input	Output	Input/Output
JMAG_RT_IM	• Tm: Load torque (N*m) • switch signal A: Switch signal of A-phase (On: 1, Off: 0) • switch signal B: Switch signal of B-phase (On: 1, Off: 0) • switch signal C: Switch signal of C-phase (On: 1, Off: 0)	• is_abc: Three-phase current (ia, ib, ic) (A) • is_qd: dq-axis current (is_q, is_d) (A) • vs_qd: dq-axis voltage (is_q, is_d) (V) • wm: Rotation speed (rad/s) • thetam: Rotation angle (rad) • Te: Output torque (N*m)	• A: A-phase terminal • B: B-phase terminal • C: C-phase terminal



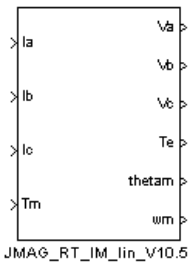
■ Voltage signal input

Block	Input	Output	Input/Output
JMAG_RT_IM_Vin	• Tm: Load torque (N*m) • Va: Electric potential of U-phase terminal (V) • Vb: Electric potential of V-phase terminal (V) • Vc: Electric potential of W-phase terminal (V)	• is_abc: Coil current (ia, ib, ic) (A) • thetam: Rotation angle (rad) • wm: Rotation speed (rad/s) • Te: Output torque (N*m)	-



■ Current signal input

Block	Input	Output	Input/Output
JMAG_RT_IM_Iin	• Tm: Load torque (N*m) • Ia: Current of U-phase coils (A) • Ib: Current of V-phase coils (A) • Ic: Current of W-phase coils (A)	• thetam: Rotation angle (rad) • wm: Rotation speed (rad/s) • Te: Output torque (N*m)	-



13-14. Creating a Model for JMAG-RT Using the Command Line

The input data (base model) for creating a JMAG-RT model can be created using the command line instead of using the JMAG-RT Library Manager.

The command line is useful to create multiple base models simultaneously.

1. Start JMAG-RT Library Manager, and then create a base model.

▶▶ Reference

For information about creating model data, see steps 2 through 4 in P.1021 "13-4. Using JMAG-RT (JMAG-RT Library Manager)".

The settings of the base model are used as the default value of each parameter for a new base model. However, this procedure is not necessary if a model is created using an existing base model, or if the default values specified in the JMAG-RT Library Manager are used.

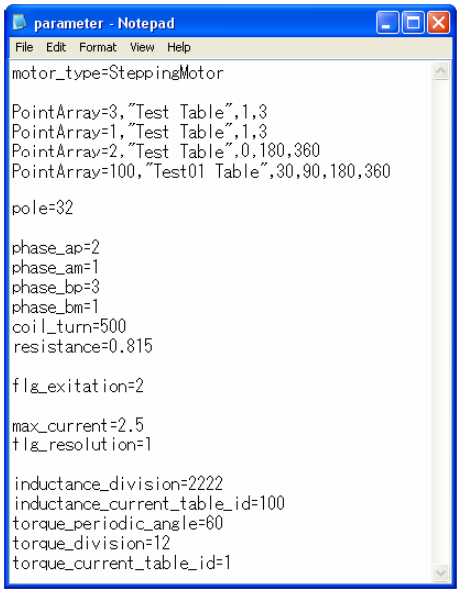
2. Create a parameter file (*.txt).

Include the changes for each parameter of the base model created in step 1.

The following should be noted when creating a parameter file.

- The text format (*.txt) must be used.
- The name of the file can be specified arbitrarily.
- A parameter file needs to be created for each calculation.
- The parameter and the value are connected with an equal sign (=).

An example is indicated in the figure below.



Example

Reference

For more information about each parameter., refer to the following pages.

- P.1153 "13-14-1. Defining a Point Sequence (PointArray)"
- P.1154 "13-14-2. Parameters for the Three-phase PM Synchronous Motor (PMSMotor)"
- P.1159 "13-14-3. Parameters for the Stepping Motor (SteppingMotor)"
- P.1161 "13-14-4. Parameters for Linear Solenoid (Linear-Solenoid)"
- P.1163 "13-14-5. Parameters for the Linear Synchronous Motor (LSMotor)".
- P.1166 "13-14-6. Parameters for the SR Motor (SRMotor)"
- P.1168 "13-14-7. Parameters for the Induction Motor (InductionMotor)"

3. Start the command prompt, specify, and then execute the commands.

Alternatively, create a batch file (*.bat) containing the commands, and then execute the file.

Specify one of the following commands.

■ Structure 1

Use the structure below to create a new base model using an existing base model.

```
\>"(full path to RTSimulateRunner.exe)\RTSimulateRunner.exe"
-model="path of the model directory containing the base model
to reference"
-file="full path to the parameter file (*.txt)"
-exec= Flag for creating a model. If '-exec' is used, a JMAG-
RT model is created after the base model is created.
```

Example:

```
\>"C:\Program Files\JMAG-
Designer12.0\JMAG_RT\RTSimulateRunner.exe"
-model="D:\StudioData\Model"
-file ="D:\StudioData\FEM\Test.txt"
-exec
```

The model directory is the directory that can be set in the [Information] tab of the JMAG-RT Library Manager. A new base model is created using a series of input data and the parameter file located in the model directory with this structure.

An example of a model directory is follows.

```
C:\Documents and Settings\(Login name)\MyDocuments\JMAG
\JMAG_RT\RT_DATAXXXX
```

MEMO The command should not be executed while the JMAG-RT Library Manger is opened when "-exec" is used.

MEMO Specify the commands above in a text editor, specify a name, and then save to use a batch file. Change the extension of the file to (*.bat), and then double-click the file. The commands are executed.

Important **A command and an argument must be included on one line. Enter "^" (hat) at the end of a line to use two lines.**

```
\>"C:\Program Files\JMAG-Designer12.0\JMAG_RT^
\RTSimulateRunner.exe"
```

■ Structure 2

Use the structure below to create new input data using the default values set in JMAG-RT Library Manager.

```
\>"(full path to RTSimulateRunner.exe)\RTSimulateRunner.exe"
-file="\full path to the parameter file"
-jcf="full path to the JCF file containing the FEM model to
use for calculation"
-exec=Flag for creating a model. If '-exec' is used, JMAG-RT
model is created soon after the base model is created.
```

Example:

```
\>"C:\Program Files\JMAG-
Designer12.0\JMAG_RT\RTSimulateRunner.exe"
-file="D:\StudioData\Parameter\parameter.txt"
-jcf="D:\StudioData\FEM\Test.jcf"
-exec
```

The JCF file of an FEM model is the JCF file specified in the [JCF File] box of the [Model Type] dialog box that is displayed by clicking [New] in JMAG-RT Library Manager.

A base model is created using the JCF file, parameter file, and default values specified in the JMAG-RT Library Manager with this structure.

MEMO The command should not be executed while the JMAG-RT Library Manger is opened when "-exec" is used.

MEMO Specify the commands above in a text editor, specify a name, and then save to use a batch file. Change the extension of the file to (*.bat), and then double-click the file. The commands are executed.

Important **A command and an argument must be included on one line. Enter "^" (hat) at the end of a line to use two lines.**

```
\>"C:\Program Files\JMAG-Designer12.0\JMAG_RT^
\RTSimulateRunner.exe"
```

When the commands are executed, a base model is created. If the JMAG-RT Library Manager is running, the base model that is created is automatically added to the [RT Model Library] list.

To run the analysis, follow the procedure from Step 5 in P.1021 "13-4. Using JMAG-RT (JMAG-RT Library Manager)".

Important **If the JMAG-RT Library Manger is running when a command including "-exec" is executed, the JMAG-RT model is not created after the base model is created.**

13-14-1. Defining a Point Sequence (PointArray)

A point sequence that is referenced by each parameter can be defined.

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
id	Integer		Point sequence ID (positive integer) The specified point sequence ID is referenced by each parameter.
name	String		Name of the point sequence The name should be enclosed in double quotation marks.
value	Double		Each value of point sequence

■ **Example:**

```
PointArray=1,"Undefined",1.0,2.0,3.0
```

13-14-2. Parameters for the Three-phase PM Synchronous Motor (PMSMotor)

The parameters that can be specified for a three-phase PM synchronous motor are as follows.

▶▶ Reference

For more information about each of the parameters, see P.1033 "13-6. Creating JMAG-RT Model from a CSV File".

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
motor_type	Flag	PMSMotor	Model type: Three-phase PM synchronous motor
accuracy_type	Flag	1 3 4	Motor model type: • Ld,Lq model • Spatial harmonic model • Ld,Lq model + Spatial harmonic model
pole	Integer		Number of poles
phase_u	Integer		FEM coil: U-phase
phase_v	Integer		FEM coil: V-phase
phase_w	Integer		FEM coil: W-phase
coil_turn	Double		Number of coil turns
flg_winding_pattern	Flag	0 1	Connection pattern: • Y-connection • Delta-connection
resistance	Double		Phase resistance (ohm)
flg_drive	Flag	0 1	Drive type: • Sinusoidal • Other
flg_resolution	Flag	0 1	Current: • Simple setting (A) • Current phase range (deg)
rated_current	Double		Rated current (A) Available if flg_resolution=0
phase_range_start	Double		Start angle for the current phase range (deg) Available if flg_resolution=0

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
phase_range_end	Double		End angle for the current phase range (deg) Available if flg_resolution=0
flg_half_periodic	Flag	0 1	Inductance Available if accuracy_type=3 • No periodicity for 1/2 electric angle • Periodicity for 1/2 electric angle
inductance_current_table_id	Integer		Point sequence ID for the current amplitude of inductance
inductance_beta_table_id	Integer		Point sequence ID for the current phase of inductance Available if accuracy_type=1 or 3
inductance_angle_resolution	Double		Mechanical angle resolution for inductance (deg) Available if accuracy_type=2 or 3
flg_torque_setting	Flag	0 1	Available if accuracy_type=3 • Do not use the same settings between inductance and torque. • Use the same settings for both inductance and torque.
torque_current_table_id	Integer		Point sequence ID for the current amplitude of torque Available if accuracy_type=2 or 3
torque_beta_table_id	Integer		Point sequence ID for the current phase of torque Available if accuracy_type=2 or 3
torque_angle_resolution	Double		Mechanical angle resolution for torque Available if accuracy_type=2 or 3

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
flg_auto_calc	Flag	0 1	Type of periodic angle for torque • Do not calculate automatically for one periodic angle from the number of poles • Calculate automatically for one periodic angle from the number of poles
periodic_angle	Double		Periodic angle for torque (deg)
flg_resolution_ldlq	Flag	0 1	LdLq Current • Simple Settings • Advanced Settings
rated_current_ldlq	Double		LdLq Rated Current (A) Available if flg_resolution_ldlq =0
phase_range_start_ldlq	Double		Start angle for the LdLq current phase range (deg) Available if flg_resolution_ldlq =0
phase_range_end_ldlq	Double		End angle for the LdLq current phase range (deg) Available if flg_resolution=0
inductance_current_ldlq_table_id	Int		Point sequence ID for the current amplitude of inductance for LdLq Available if accuracy_type=1 or 4
inductance_beta_ldlq_table_id	Int		Point sequence ID for the current phase of inductance for LdLq Available if accuracy_type=1 or 4

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
flg_iron_loss	Flag	0 1	Consider iron loss • Not considered • Considered
flg_iron_loss_resolution	Flag	0 1	Consider iron loss • Simple Settings • Advanced Settings
max_rotate_velocity	Double		Maximum rotational speed (rpm)
rotate_velocity_table_id	Int		Point sequence ID for rotational speed Available if flg_iron_loss_resolution=1
coef_magnet_temperature	Double		Temperature correction coefficient of magnet (%/K)
base_magnet_temperature	Double		Base temperature for magnet (K)
coef_coil_temperature	Double		Temperature correction coefficient of coil (ppm/K)
base_coil_temperature	Double		Base temperature for coil (K)
inertia_correction_factor1	Double		Correction coefficient (coef_inertia1) of α inertia moment
inertia	Double		J0 moment (MomentofInertia (Kgm2))

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
inertia_correction_factor2	Double		Correction coefficient (coef_inertia1) of β inertia moment
time_constant	Double		Time constant Tmech (MechTimeConstant(sec))
flg_remote	Flag	0 1	Distributed calculation: • Do not distribute • Distribute
cpu_group_name	String		CPU group name Server: Port ID: Group name For example: 10.157.101.2:8888:test1
flg_recalculation	Flag	0 1 2	Calculation mode: • Recalculate only the failed cases • Recalculate all cases • Recalculate all cases with increased resolution
flg_plot_file	Flag	0 1	Control result files: • Export the distribution amount into result files • Do not export the distribution amount into result files
flg_delete_result	Flag	0 1	Delete result files after calculation. • Leave result files after calculation • Delete result files after calculation

13-14-3. Parameters for the Stepping Motor (SteppingMotor)

The parameters that can be specified for a stepping motor are as follows.

▶▶ Reference

For more information about specifying each parameter, see P.1073 "13-9. Two-phase Stepping Motor".

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
motor_type	Flag	SteppingMotor	Model type: Two-phase stepping motor
pole	Integer		Number of poles
phase_ap	Integer		FEM coil: A-phase
phase_am	Integer		FEM coil: -A-phase
phase_bp	Integer		FEM coil: B-phase
phase_bm	Integer		FEM coil: -B-phase
coil_turn	Integer		Number of turns
resistance	Double		Phase resistance (ohm)
flg_exitation	Flag	1 2 3	Excitation: • One phase excitation • One-two excitation • Two phase excitation
max_current	Double		Upper limit of the current range
flg_resolution	Flag	0 1	Resolution • Disable the resolution setting • Enable the resolution setting
inductance_division	Integer		Number of divisions for the rotation angle for inductance
inductance_current_table_id	Integer		Point sequence ID the current versus inductance during excitation
torque_periodic_angle	Double		Rotation angle for torque (deg)
torque_division	Integer		Number of divisions for the rotation angle for torque

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
torque_current_table_id	Integer		Point sequence ID for the current versus torque during excitation
inertia_correction_factor1	Double		Correction coefficient (coef_inertia1) of inertia moment
inertia	Double		Inertia moment J0 (Kgm2)
inertia_correction_factor2	Double		Correction coefficient (coef_inertia2) of inertia moment
time_constant	Double		Time constant t-mech (sec)
flg_remote	Flag	0 1	Distributed calculation • Do not distribute • Distribute
cpu_group_name	String		CPU group name Server: Port ID: Group name For example: 10.157.101.2:8888:test1
flg_recalculation	Flag	0 1 2	Calculation mode • Recalculate only the failed cases • Recalculate all cases • Recalculate all cases with increased resolution
flg_plot_file	Flag	0 1	Control result files: • Export the distribution amount into result files • Do not export the distribution amount into result files
flg_delete_result	Flag	0 1	Delete result files after calculation. • Leave result files after calculation • Delete result files after calculation

13-14-4. Parameters for Linear Solenoid (LinearSolenoid)

The parameters that can be specified for a linear solenoid motor are as follows.

▶▶ Reference

For more information about specifying each parameter, see P.1090 "13-10. Linear Solenoid".

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
motor_type	Flag	LinearSolenoid	Model type: Linear solenoid
phase	Integer		FEM coil ID
coil_turn	Double		Number of coil turns
resistance	Double		Phase resistance (ohm)
max_position	Double		Upper limit of motion range (m)
min_position	Double		Lower limit of motion range (m)
position_division	Integer		Number of divisions for resolution (position)
flg_motion_table	Flag	0 1	Specify the point sequence for the resolution (position) • Set automatically based on the upper/lower limit of motion range and the number of divisions • Specify a point sequence
motion_table_id	Integer		Point sequence ID for the resolution (position)
current_division	Integer		Number of divisions for the resolution (current)
max_current	Double		Maximum value of the current range (A) for the resolution (current)
flg_current_table	Flag	0 1	Point sequence for the resolution (current) • Set automatically based on the current range and the number of divisions • Specify a point sequence

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
current_table_id	Integer		Point sequence ID for the resolution (current)
mass_factor1	Double		Mass correction coefficient (coef_mass1)
mass	Double		Mass(M0)(Kg)
mass_factor2	Double		Mass correction coefficient (coef_mass2)
time_constant	Double		Time constant Tmech (MechTimeConstant (sec))
flg_remote	Flag	0 1	Distributed calculation • Do not distribute • Distribute
cpu_group_name	String		CPU group name Server: Port ID: Group name For example: 10.157.101.2:8888:test1
flg_recalculation	Flag	0 1 2	Calculation mode • Recalculate only the failed cases • Recalculate all cases • Recalculate all cases with increased resolution
flg_plot_file	Flag	0 1	Control result files: • Export the distribution amount into result files • Do not export the distribution amount into result files
flg_delete_result	Flag	0 1	Delete result files after calculation. • Leave result files after calculation • Delete result files after calculation

13-14-5. Parameters for the Linear Synchronous Motor (LSMotor)

The parameters that can be specified for a linear synchronous motor are as follows.

▶▶ Reference

For more information about specifying each parameter, see P.1104 "13-11. Linear Synchronous Motor".

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
motor_type	Flag	LSMotor	Model type: Linear synchronous motor
periodic_position	Double		Displacement for one period of electric angle (m)
phase_u	Integer		FEM coil: U-phase
phase_v	Integer		FEM coil: V-phase
phase_w	Integer		FEM coil: W-phase
coil_turn	Double		Number of coil turns
flg_winding_pattern	Flag	0 1	Connection pattern: • Y connection • Delta connection
resistance	Double		Phase resistance (ohm)
flg_drive	Flag	0 1	Drive type: • Sinusoidal • Other
flg_force	Flag	0 1	Attractive force • Do not export attractive force • Export attractive force
flg_force_direction	Flag	1 2 3	Direction of attractive force • X • Y • Z

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
current_amplitude_range	Double		Current amplitude range (A)
current_phase_range	Double		Current phase range (deg)
flg_resolution	Flag	0 1	Resolution • Disable the resolution setting • Enable the resolution setting
inductance_division	Integer		Inductance Number of divisions for the displacement
inductance_current_table_id	Integer		Point sequence ID for the current versus inductance during excitation
displacement_range	Double		Displacement of electromagnetic force (m)
position_division	Integer		Number of divisions for the displacement of electromagnetic force
force_current_table_id	Integer		Point sequence ID for the current versus electromagnetic force during excitation
force_beta_table_id	Integer		Point sequence ID for the current versus electromagnetic force during excitation
mass_factor1	Double		Mass correction coefficient (coef_mass1)
mass	Double		Mass (M0)(Kg)

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
mass_factor2	Double		Mass correction coefficient (coef_mass2)
time_constant	Double		Time constant τ-mech (sec)
flg_remote	Flag	0 1	Distributed calculation • Do not distribute • Distribute
cpu_group_name	String		CPU group name Server: Port ID: Group name For example: 10.157.101.2:8888:test1
flg_recalculation	Flag	0 1 2	Calculation mode • Recalculate only the failed cases • Recalculate all cases • Recalculate all cases with increased resolution
flg_plot_file	Flag	0 1	Control result files: • Export the distribution amount into result files • Do not export the distribution amount into result files
flg_delete_result	Flag	0 1	Delete result files after calculation. • Leave result files after calculation • Delete result files after calculation

13-14-6. Parameters for the SR Motor (SRMotor)

The parameters that can be specified for a SR motor are as follows.

▶▶ Reference

For more information about specifying each parameter, see P.1126 "13-12. SR Motor".

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
motor_type	Flag	SRMotor	Model type: SR motor
pole	Integer		Number of poles
phase	Integer		Number of phases
phase_A	Integer		FEM coil: A-phase
phase_B	Integer		FEM coil: B-phase
phase_C	Integer		FEM coil: C-phase
phase_D	Integer		FEM coil: D-phase
phase_E	Integer		FEM coil: E-phase
coil_turn	Integer		Number of turns
resistance	Double		Phase resistance (ohm)
flg_resolution	Flag	0 1	Resolution • Disable the resolution setting • Enable the resolution setting
inductance_division	Integer		Number of divisions for the rotation angle for inductance
inductance_current_table_id	Integer		Point sequence ID the current versus inductance during excitation
rated_current	Double		Rated current (A) Available if resolution=0
torque_current_table_id	Integer		Point sequence ID for the current versus torque during excitation
torque_angle_resolution	Double		Mechanical angle resolution for torque Available if accuracy_type=2 or 3

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
flg_auto_calc	Flag	0 1	Type of periodic angle for torque • Do not calculate automatically for one periodic angle from the number of poles • Calculate automatically for one periodic angle from the number of poles
periodic_angle	Double		Periodic angle for torque (deg)
flg_remote	Flag	0 1	Distributed calculation • Do not distribute • Distribute
cpu_group_name	String		CPU group name Server: Port ID: Group name For example: 10.157.101.2:8888:test1
flg_recalculation	Flag	0 1 2	Calculation mode • Recalculate only the failed cases • Recalculate all cases • Recalculate all cases with increased resolution
flg_plot_file	Flag	0 1	Control result files: • Export the distribution amount into result files • Do not export the distribution amount into result files
flg_delete_result	Flag	0 1	Delete result files after calculation. • Leave result files after calculation • Delete result files after calculation

13-14-7. Parameters for the Induction Motor (InductionMotor)

The parameters that can be specified for a induction motor are as follows.

Reference

For more information about each of the parameters, see P.1139 "13-13. Induction Motor".

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
motor_type	Flag	InductionMotor	Model type: Induction motor
pole	Integer		Number of poles
model_division	Integer		Number of model division
phase_u	Integer		FEM coil: U-phase
phase_v	Integer		FEM coil: V-phase
phase_w	Integer		FEM coil: W-phase
coil_turn	Double		Number of coil turns
resistance	Double		Phase resistance (ohm)
rated_frequency	Double		Rated frequency (Hz)
rated_voltage	Double		Rated voltage (V)
flg_iron_loss	Flag	0 1	Consider iron loss • Not considered • Considered
flg_remote	Flag	0 1	Distributed calculation: • Do not distribute • Distribute

Property (propName)	Type (Type)	Flag (scriptKeyName)	Description
cpu_group_name	String		CPU group name Server: Port ID: Group name For example: 10.157.101.2:8888:test1
flg_recalculation	Flag	0 1 2	Calculation mode: • Recalculate only the failed cases • Recalculate all cases • Recalculate all cases with increased resolution
flg_plot_file	Flag	0 1	Control result files: • Export the distribution amount into result files • Do not export the distribution amount into result files
flg_delete_result	Flag	0 1	Delete result files after calculation. • Leave result files after calculation • Delete result files after calculation

13-15. Running Simulations Using Programs in C

13-15-1. Flow of Simulations Using Programs in C

A simulation using a program in C can be run as follows.

1. **Create a JMAG-RT model using the JMAG-RT Library Manager.**

   ▶▶ Reference

   For information about creating a JMAG-RT model, seeP.1021 "13-4. Using JMAG-RT (JMAG-RT Library Manager)".

2. **Create a program in C.**

   ▶▶ Reference

   For information about the structure and functions of programs in C, see P.1172 "13-15-2. Simulation of Programs in C Using JMAGRT_DLL.dll", P.1173 "13-15-3. Functions in JMAGRT_DLL.dll".

3. **Open the program in C that is created in a text editor such as Note Pad, and then specify the number of steps, JMAG-RT model data file (*.rtt) to reference, drive type, and default values for the JMAG-RT model.**

4. Build the program in C to create an execution file (*.exe).

▶▶ Reference

For information about building and debugging programs in C, see P.1182 "13-15-5. Building a Program in C" and P.1183 "13-15-6. Debugging a Program in C".

5. Place all the files required for a simulation in the same folder.

- The JMAG-RT model data file (*.rtt) created in Step 1
- The execution file (*.exe) created in Step 4
- The JMAGRT_DLL.dll file (*.dll)

These files should be saved in the following directory. Use a copy of these files.

```
(JMAG installation directory)\JMAG_RT\general
```

The default directory that these files are stored is as follows:

```
C:\Program Files\JMAG-Studio (or JMAG-Designer)\JMAG_RT
\general
```

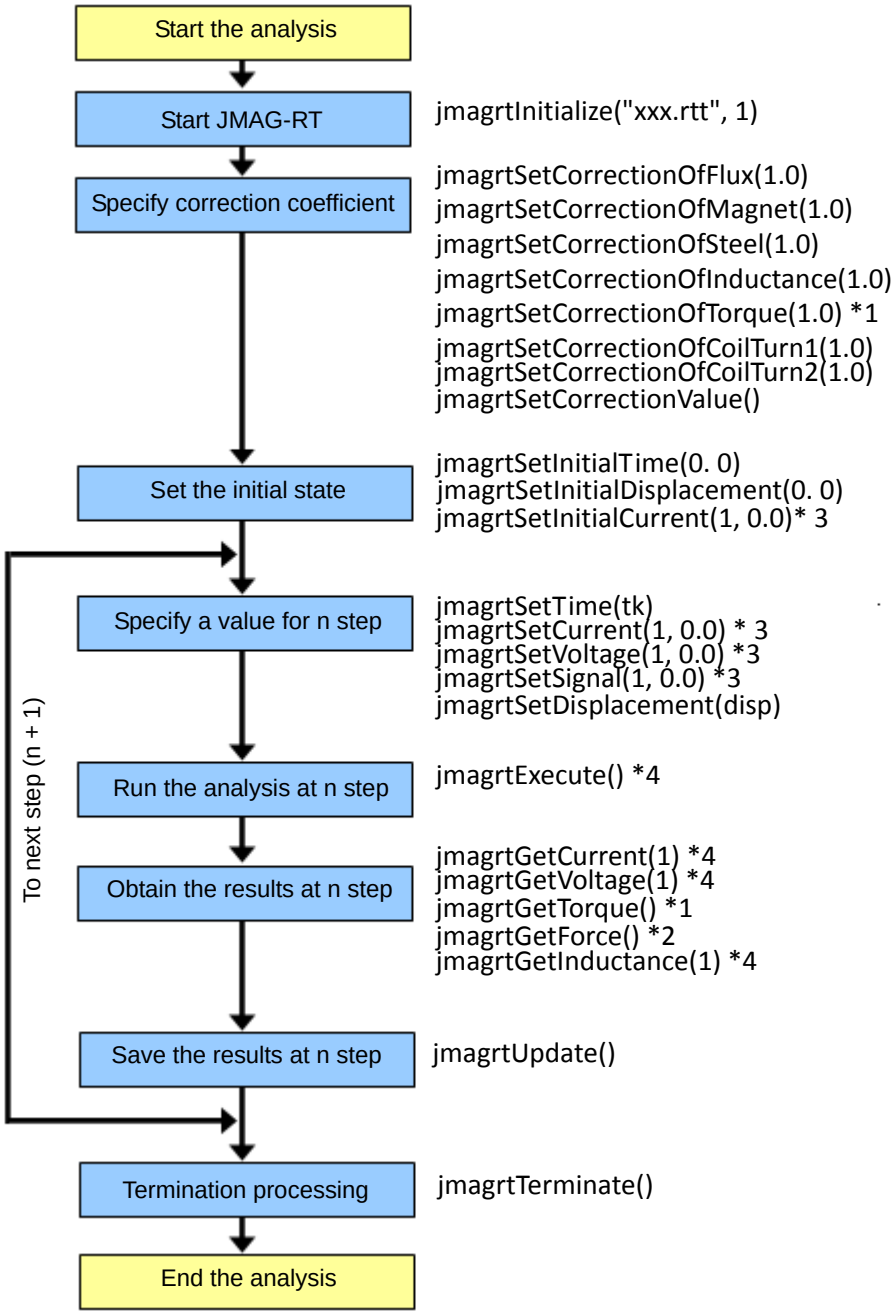
Important **The JMAGRT_DLL.dll file may be updated when JMAG is updated. If JMAG is updated with a service pack, the JMAGRT_DLL.dll file located in the directory described above needs to be re-copied.**

6. Execute the execution file (*.exe)

A simulation starts running.

13-15-2. Simulation of Programs in C Using JMAGRT_DLL.dll

A program in C needs to have the following structure.



*1: Can be used for a three-phase PM synchronous motor and two-phase stepping motor.

*2: Can be used for a linear solenoid and linear synchronous motor.

*3: Specify only the number of terminals.

*4: Can only be used for an electric signal type.

13-15-3. Functions in JMAGRT_DLL.dll

The following functions must be specified.

- jmagrtInitialize
  Initialize the program
- jmagrtSetCorrection***
  Specify the correction value for the magnetic flux and magnets
- jmagrtUpdate
  Save the data for the current step
- jmagrtTerminate
  Terminate the program

The order of the following functions cannot be changed.

1. **jmagrtInitialize**
   Initializes the program
2. **jmagrtSetCorrection*****
   Specifies the correction value for the magnetic flux and magnets
3. **jmagrtSetInitial*****
   Specifies the default values of each physical quantity such as current and voltage.

■ Functions to start JMAG-RT

```
jmagrtInitialize(char*rtt_file_name, int interface_type)
```

Function	Specifies the path to the JMAG-RT model data file and the drive type
Parameter	rrt_file_name: Path to the JMAG-RT model data file (*.rtt) Interface_type: Drive type 1: Electric signal input, 2: Current signal input
Return value	None
Description	This command must be specified. The following processes are executed by this command. 1. Obtains the JMAG-RT model data file (rrt_file_name) and drive type (Interface_type) as a parameter. 2. Obtains the motor type (ModelType) from the JMAG-RT model data file. 3 Determines the number of input terminals from the motor type and drive type. 4. Creates an object (Calc_Eq) based on the motor type.

■ Functions to set correction coefficient

```
jmagrtSetCorrectionOfFlux(double value)
```

Function	Specifies the correction coefficient for linkage flux of a magnet.
Parameter	value: Correction value for linkage flux of a magnet The default value is 1.0.
Return value	None

```
jmagrtSetCorrectionOfMagnet(double value)
```

Function	Corrects the grade of a magnet.
Parameter	value: Correction value for the grade of a magnet. The correction coefficient is used to take into account the effects when a magnet is changed. This coefficient affects the linkage flux of a magnet, torque, and electromagnetic force. The default value is 1.0.
Return value	None

`jmagrtSetCorrectionOfSteel(double value)`

Function	Corrects the grade of a steel sheet.
Parameter	value: Correction value for the grade of a steel sheet. The correction coefficient is used to take into account the effects when a material is changed. This coefficient affects the linkage flux of a magnet, torque, and electromagnetic force. The default value is 1.0.
Return value	None

`jmagrtSetCorrectionOfInductance(double value)`

Function	Specifies the correction coefficient for inductance.
Parameter	value: Correction coefficient for inductance. The default value is 1.0.
Return value	None

`jmagrtSetCorrectionOfTorque(double value)`

Function	Specifies the correction coefficient for torque.
Parameter	value: Correction coefficient for torque. The default value is 1.0.
Return value	None
Description	This command can be used for a three-phase PM synchronous motor and two-phase stepping motor.

`jmagrtSetCorrectionOfForce(double value)`

Function	Specifies the correction coefficient for electromagnetic force.
Parameter	value: Correction coefficient for electromagnetic force. The default value is 1.0.
Return value	None
Description	This command can be used for a linear solenoid and linear synchronous motor.

`jmagrtSetCorrectionOfCoilTurn1(double value)`

Function	Specifies the correction coefficient (1) for the number of coil turns.
Parameter	value: Correction coefficient (1) for the number of coil turns. The default value is 1.0. With this setting, the effects of changing the number of turns is taken into account. This value affects the inductance, linkage flux of a magnet, and electromagnetic force. If the number of turns does not need to be corrected, specify the number of turns of the original motor.
Return value	None

`jmagrtSetCorrectionOfCoilTurn2(double value)`

Function	Specifies the correction coefficient (2) for the number of coil turns.
Parameter	value: Correction coefficient (2) for the number of coil turns. The default value is 1.0. With this setting, the effects of changing the number of turns is taken into account. This value affects the inductance, linkage flux of a magnet, and electromagnetic force. If the number of turns does not need to be corrected, specify the number of turns of the original motor.
Return value	None

`jmagrtSetCorrectionValue()`

Function	Applies each correction value to a model.
Parameter	None
Return value	None
Description	Even if the correction coefficients are not used, this function is necessary.

■ Functions to set initial status of a model

`jmagrtSetInitialTime(double time)`

Function	Specifies the start time of an analysis.
Parameter	time: Initial time (sec) The default value is 0.0.
Return value	None

`jmagrtSetInitialCurrent(int terminal_id, double current)`

Function	Specifies the current value at the start time of an analysis.
Parameter	terminal_id: Terminal ID. Enter '1' for setting the terminal 1 current: Initial current (A) The default value is 0.0.
Return value	None
Description	This command needs to be set for each terminal.

`jmagrtSetInitialDisplacement (double disp)`

Function	Specifies the position of a mover at the start time of an analysis with the angle or distance.
Parameter	disp: Angle (deg) or distance (m) The default value is 0.0.
Return value	None
Description	This command specifies the initial angle for a three-phase PM synchronous motor and two-phase stepping motor. This command specifies the initial position for a linear solenoid and linear synchronous motor.

■ Functions to set the input value for each step

`jmagrtSetTime(double time)`

Function	Specifies the time for each step.
Parameter	time: Time (sec) The default value is 0.0.
Return value	None

`jmagrtSetCurrent(int terminal_id, double current)`

Function	Specifies the current value of a terminal for each step
Parameter	terminal_id: Terminal ID Enter '1' for setting the terminal 1. current: Current value (A) The default value is 0.0.
Return value	None
Description	This command needs to be set for each terminal

`jmagrtSetVoltage(int terminal_id, double voltage)`

Function	Specifies the voltage of a terminal for each step.
Parameter	terminal_id: Terminal ID Enter '1' for setting the terminal 1. voltage: Specifies voltage between the specified terminal and ground component of a three-phase PM synchronous motor and linear synchronous motor. Specifies the voltage of a terminal for a two-phase stepping motor and linear solenoid.
Return value	None
Description	This command needs to be set for each terminal.

`jmagrtSetSignal(int terminal_id, int signal)`

Function	Specifies the current flow status of a terminal for each step.
Parameter	terminal_id: Terminal ID. Enter '1' for setting the terminal 1 signal=0: Switch is open. signal=1: The current is flowing.
Return value	None
Description	This command needs to be set for each terminal.

```
jmagrtSetDisplacement(double disp)
```

Function	Specifies the position of a mover for each step with the angle or distance.
Parameter	disp: Angle (deg) or distance (m). The default value is 0.0.
Return value	None
Description	This command specifies the initial angle for a three-phase PM synchronous motor and two-phase stepping motor. This command specifies the initial position for a linear solenoid and linear synchronous motor. If the position is not set, the initial position specified for jmagrtSetInitialDisplacement is used.

■ Functions to run an analysis for each step

```
jmagrtExectute()
```

Function	Runs an analysis for one step to calculate the current (or voltage) and electromagnetic force (torque).
Parameter	None
Return value	None
Description	The initial position needs to be set for each step in the case of electric signal input. This setting is not necessary for a current signal input.

■ Functions to obtain the analysis results for each step

```
jmagrtGetCurrent(int terminal_id)
```

Function	Obtains the current value of the specified terminal
Parameter	terminal_id: Terminal ID Enter '1' for setting the terminal 1
Return value	Current (A)
Description	This command can be used at each step for electric signal input. This command cannot be used for a current signal input.

`jmagrtGetVoltage(int terminal_id)`

Function	Obtains the voltage of the specified terminal
Parameter	terminal_id: Terminal ID Enter '1' for setting the terminal 1
Return value	Voltage (V)
Description	This command can be used at each step for electric signal input. This command cannot be used for a current signal input.

`jmagrtGetTorque()`

Function	Obtains torque.
Parameter	None
Return value	Torque (N*m)
Description	This command can be used for a three-phase PM synchronous motor and two-phase stepping motor.

`jmagrtGetForce()`

Function	Obtains electromagnetic force.
Parameter	None
Return value	Electromagnetic force (N)
Description	This command can be used for a linear solenoid and linear synchronous motor.

`jmagrtGetInductance(int coil_id1, int coil_id2)`

Function	Obtains the inductance between two coils.
Parameter	coil_id1, coil_id2: Coil ID. Enter '1' to obtain the first coil data.
Return value	Inductance (H)
Description	This command cannot be used for electric signal input. For a three-phase PM synchronous motor and linear synchronous motor, the relation between coil_id and each phase is shown below. coil_id = 1: U-phase, coil_id= 2: V-phase, coil_id = 3: W-phase For a two-phase stepping motor, the relation between coil_id and each phase is shown below. coil_id = 1: A+ phase, coil_id= 2: A- phase, coil_id = 3: B+ phase, coil_id= 4: B- phase

■ Functions to save the analysis results of each step

`jmagrtUpdate()`

Function	Saves the data of the current step.
Parameter	None
Return value	None
Description	This command must be used to confirm the analysis at each step. When this command is executed, the following is performed. 1. Save the data of the current step as the previous data. 2. Save the previous data as the data of the pre-previous step. The following data is saved. • Time • Current • Magnetic flux • Inductance • Mechanical angle

■ Functions to terminate an analysis

`jmagrtTerminate()`

Function	Terminates any open memory.
Parameter	None
Return value	None
Description	This command must be used when an analysis is complete.

13-15-4. Samples of Programs in C

The samples of programs in C are saved in the following directory.

```
JMAG installation folder\JMAG_RT\general
```

The default directory for the samples of programs in C is as follows:

```
C:\Program Files\JMAG-Studio(or JMAG-Designer)\JMAG_RT\general
```

13-15-5. Building a Program in C

A program in C can be build and an execution file (*.exe) can be created as follows:

1. **Start Visual Studio 2005.**
2. **Select [File] > [New] from the menu bar, and then select [Win32 Console Application]. Specify a project name.**
3. **Place the JMAGRT_DLL.dll file in [Workspace Project].**
4. **Select [File] > [New] > [C++ Source File] from the menu bar, and then specify 'main' for a file name.**
   Alternatively, select [Project] > [Add to Project] > [File] from the menu bar, and then add 'main.cpp'.
5. **Create 'main.cpp'.**

   As a result, 'jam_run' is created.

13-15-6. Debugging a Program in C

A C language program can be debugged as follows.

1. **Create a debug version of an execution file (*.exe) that is created by a user.**
2. **Specify a command to start Visual Studio 2005 in the command prompt.**
3. **Specify the following command.**

   ```
   devenv (project name that is specified by user)
   ```

4. **Select [Project] > [Set].**
5. **Select a debug.**
6. **Click [OK].**
7. **Select [Build] > [Debug] > [Execute].**
8. **Debug the created program.**

13-16. Executing Analysis using Third-party Job Management Systems

JMAG-RT has the following modes built-in.

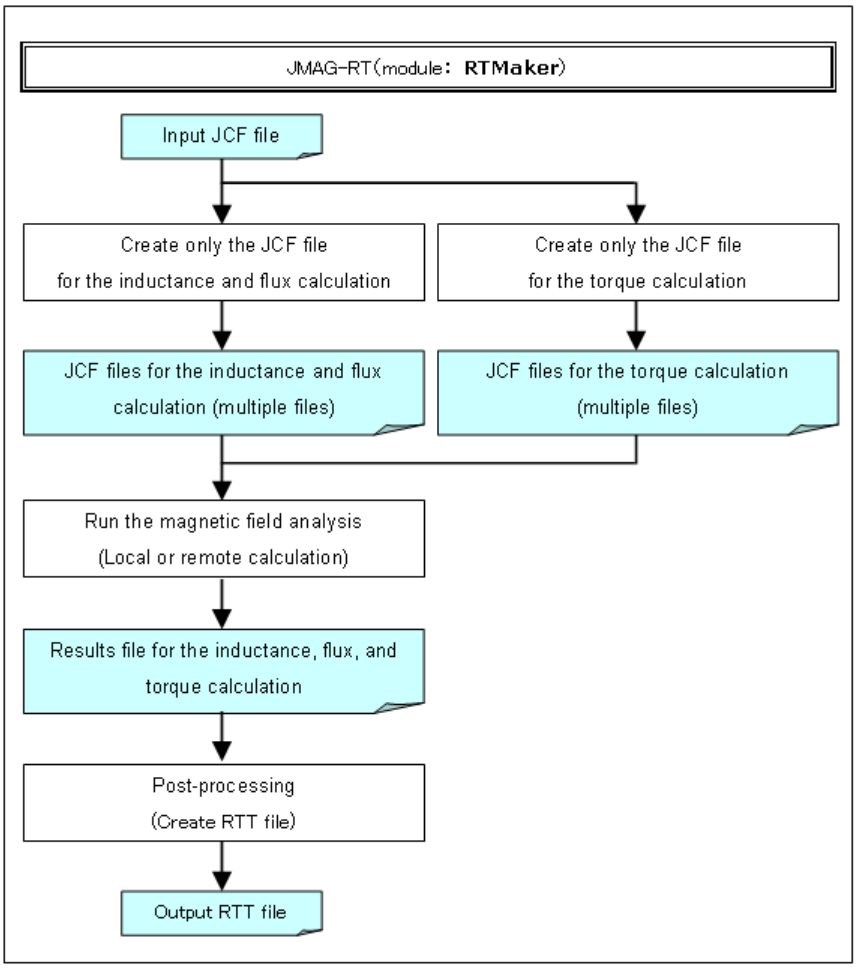
- Mode to create a JCF file for the Flux, Inductance calculation.
- Mode to create a JCF file for the Torque calculation.
- A mode for post-processing based on the analysis results (creates RTT file).

Analysis can be run and managed using a third-party job management system using these modes.

Important **Processes for forwarding input files that are generated and the result files obtained after an analysis as well as adding jobs into a job management system need to be defined by script created by the user.**

■ Call flow

The call flow for job management systems are as follows:


JMAG-RT(module: RTMaker)
Input JCF file
Create only the JCF file for the inductance and flux calculation
Create only the JCF file for the torque calculation
JCF files for the inductance and flux calculation (multiple files)
JCF files for the torque calculation (multiple files)
Run the magnetic field analysis (Local or remote calculation)
Results file for the inductance, flux, and torque calculation
Post-processing (Create RTT file)
Output RTT file

■ Call command

The following command is executed from the command line to run JMAG-RT (module: RTMaker) in a job management system.

```
./RTMaker.exe
```

In addition, the following argument is required after the command.

- "-in=[input JCF file name (absolute path)]"
- "-xml=rtmotor.xml"

MEMO The rtmotor.xml creates a folder to save the data when a JMAG-RT model is created using the JMAG-RT Library Manager. The parameters required to create a JMAG-RT model are included in this file in the XML format.

In addition, the following argument needs to be added depending on the model.

- Mode to create a JCF file for the Flux, Inductance calculation
  Argument: "-induct_jcf=yes"
- Mode to create a JCF file for the Torque calculation
  Argument: "-torque_jcf=yes"
- Mode for post-processing (create RTT file)
  Argument: "-post=yes"

MEMO The mode to run the magnetic field analysis is run using only arguments for the JCF file name and XML name.

Chapter 14. JMAG-RT Viewer

This chapter describes the JMAG-RT Viewer which one of the tools included with JMAG.

▶▶ Reference

For information on JMAG-RT, see "Chapter 13. JMAG-RT" on page 1015.

14-1. About JMAG-RT Viewer

JMAG-RT Viewer has following three main features.

■ Display the motor information

The JMAG-RT model is a motor behavior model which is created using JMAG-RT or JMAG-Express and is contained in an RTT file. The RTT file contains the motor characteristics reflecting the geometry and the material properties obtained by the FEM analysis. Using the model as a component block in a circuit/control simulator, such as Simulink or PSIM, the simulator can take into account the motor characteristics.

The circuit simulators can only display part of the motor information contained in the JMAG-RT file. The JMAG-RT Viewer, can be used to display the all information about the model. For just displaying the basic motor information, no extra license is required.

▶▶ Reference

- For information about JMAG-RT, see "Chapter 13. JMAG-RT" on page 1015.
- For an introduction to JMAG-Express, see the JMAG-Express WEB site: http://www.jmag-international.com/express/

■ Display the efficiency map for the JMAG-RT model

The JMAG-RT Viewer can draw the efficiency map for user specified motor control technique and current and voltage limits. In addition, the iron loss map (N-T-Iron loss map) and the coil loss map (N-T-Coil loss map) also can be drawn.

■ Display the Torque-Speed curve for JMAG-RT model

The JMAG-RT Viewer can display the Torque-Speed for user specified motor control technique and current and voltage limits. In addition the d/q current-Speed (N-Id/Iq) curve and the current amplitude - Speed (N-Iam) curve also can be drawn.

MEMO Types of JMAG-RT model and accuracy of each function

The JMAG-RT model types that can be used for a three-phase PM synchronous motor are a Ld, Lq model and a spatial harmonics model.

The accuracy in inductance which is exported to a JMAG-RT model file (*.rtt) and the calculation time to create a JMAG-RT model differs depending on the type of JMAG-RT model.

Also the depending on the type of model, the shape for the inductance information used in the torque-speed curve or efficiency map differs. The corresponding components between the shape for inductance information and the types of model are described below.

	Ld, Lq Model	Spatial harmonics model	Ld, Lq model + Spatial harmonic model
The shape for inductance information used in a torque-speed curve or efficiency map	Distribution information dependent on electric current and phase	Constant value that is averaged for electric current, phase, and mechanical angle	Distribution information dependent on electric current and phase

*The [Ld, Lq model + Spatial harmonic model] displayed in the table above is when the [Ld,Lq model] and the [Spatial Harmonic Model] are both selected in the [Select Motor Model] dialog box of JMAG-RT Library Manager.

▶▶ Reference
For more information, see "13-4. Using JMAG-RT (JMAG-RT Library Manager)" on page 1021.

14-2. License for JMAG-RT Viewer

The features of the JMAG-RT Viewer are licensed in different ways.

- No license is required to display the basic motor information.
- A free license is required separately through the JSOL website when rendering torque-speed curve, Ld map, and Lq map.

  JSOL JMAG-RT Viewer web site: http://www.jmag-international.com/rt/viewer/index.html

  However, a new license does not need to be obtained if a license for rendering efficiency maps (JMAG-RT-EFFICIENCYMAP) is installed, in which these functions can be used.
- To display the efficiency map, an extra license using the FLEXlm license system with the feature key JMAG-RT-EFFICIENCYMAP is required. To obtain it, please contact your distributor or sales representative.

14-3. Notes Regarding the JMAG-RT Model Data File

- The information displayed in the JMAG-RT Viewer depends upon the version of JMAG-RT and JMAG-Express which was used to create the data file (*.rtt).
- The format of the JMAG-RT model data file (*.rtt) was changed in JMAG-Designer version10.5. To use an older format file with the JMAG-RT Viewer, it is required to convert it to the new format using the RT Library Manager, as described below.

  Steps to convert the format are as follows.

  i. Start the JMAG-RT Library Manager bundled in JMAG-Designer 10.5 or JMAG-Studio 10.03001 or later.

  ii. Select a JMAG-RT motor model from the [RT Model Library] list in the main dialog that is set to [Created] in the [Data Status] column and click [Modify].

  A message asking if you want to edit the current data is displayed.

  iii. Select [No] in the dialog box and dialog box to set each model type is displayed.

  iv. Confirm the information for the JMAG-RT model file displayed in the [Data Information] tab.

  v. Select [OK] and a message asking if you want to update the JMAG-RT model data file is displayed.

  vi. Select [Yes] and a report file is displayed on the screen whether or not updating the data has succeeded or not.

  Confirm the error message that is displayed explaining the error if it occurs.

  vii. The JMAG-RT model is copied to the working folder for the circuit/control simulator.

  The JMAG-RT model data file that is created to the working folder is converted to the new format.

- JMAG-Studio supports the new format of JMAG-RT model data file from version 10.03001 or later.
- JMAG-Express supports the new format of the JMAG-RT model data file from version 1.00.3001 or later.
  The JMAG-RT model data file created in the older versions of JMAG-Express cannot be converted to the newer format. To use the MAG-RT Viewer with these files, please re-calculate using JMAG-Express version1.00.3001 or later.

■ Feature to display the basic motor information

- This feature supports all JMAG-RT model data files (*.rtt) produced by JMAG-RT and JMAG-Express.

■ Feature to display Torque-speed curve or Efficiency map

- This feature only supports the JMAG-RT model data file (*.rtt) for the Three-Phase PM Synchronous Motor.
- To take the iron loss into account when drawing the efficiency map, the loss information must be present in the JMAG-RT model data file.
- To calculate the iron loss information when creating the JMAG-RT model data using JMAG-RT, check the [Check iron loss] on in the [Create] tab.
- To calculate the iron loss information when creating the JMAG-RT model data file using JMAG-Express, use the version which was released in June 2011 or later.

MEMO When the JMAG-RT model data file without the iron loss information is opened in the JMAG-RT Viewer, only the coil loss is taken into account as the loss source when calculating the torque-speed curve and the efficiency map.

MEMO The JMAG-RT model data file (*.rtt) used to draw torque-speed curves in JMAG-RT Viewer uses the [Ld, Lq model] information that is specified with the [Spatial Harmonic Model] information in the [Select Motor Model] dialog box for calculating a three phase permanent magnet synchronous motor in JMAG-RT.

▶▶ Reference

For more information about the [Select Motor Model] dialog box for calculating a three phase permanent magnet synchronous motor in JMAG-RT, see "13-4. Using JMAG-RT (JMAG-RT Library Manager)" on page 1021.

14-4. Starting JMAG-RT Viewer

After installation the JMAG-RT Viewer can be started from the Windows Start menu:

1. Select [Start] > [All Programs] > [JMAG-RT Viewer] > [JMAG-RT Viewer].

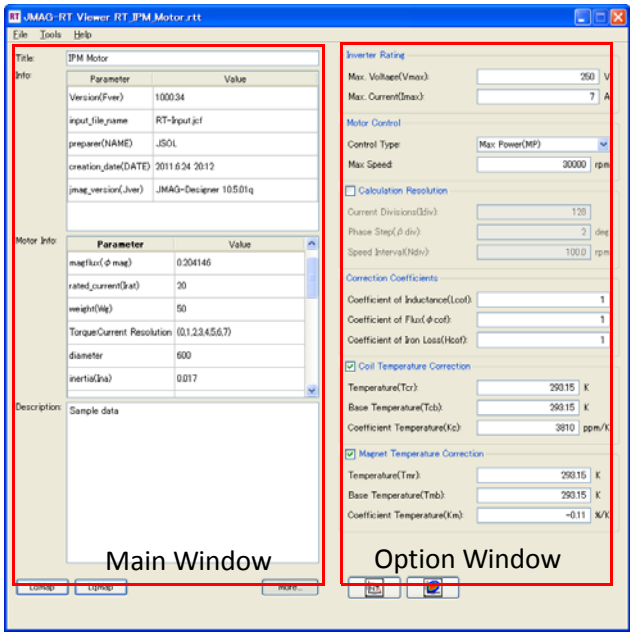
JMAG-RT Viewer window is displayed.

MEMO On installing JMAG-Designer10.5 or JMAG-Studio10.03001 or later, the JMAG-RT Viewer will be installed.

MEMO To install JMAG-RT Viewer independently, the JMAG-RT Viewer installer can be downloaded from the JSOL web site. The installation manual and the recommended operating environment are also on the web site.

14-5. Screen Configuration

The screen configuration of JMAG-RT Viewer is as follows.



14-5-1. Main Window of JMAG-RT Viewer

The main window is the one which is displayed first after the application started.

When a JMAG-RT model data file(*.rtt) is imported to the JMAG-RT Viewer, the following information in the file is displayed in the main window.

- [Title]
- [Information]
- [Motor Information]
- [Description]

MEMO JMAG-RT model data files can also be "drag and dropped" onto the main window to open them.

MEMO If the JMAG-RT model data file is output from JMAG-RT, the same information as displayed in the Information tab of the RT Library Manager is displayed.

▶▶ Reference

For steps to import the JMAG-RT model data file, see "14-7-1. Importing a JMAG-RT Model Data File" on page 1198.

And by clicking these buttons in the bottom of the main window, The Ld map and Lq map can be displayed respectively.

- [Ld map] button
- [Lq map] button

14-5-2. Detailed Settings for the Option Window

When the [More] button is clicked, the main window is expand to show the detailed setting option window. In this window, the parameters required to display the torque-speed curve and the efficiency map are set.

The following parameters can be set in the window.

- [Inverter Rating]
- [Motor Control]
- [Calculation Resolution]
- [Correction Coefficients]
- [Coil Temperature Correction]
- [Magnet Temperature Correction]

MEMO When there is no valid license for the torque-speed curve or efficiency map, the [More] button is disabled.

▶▶ Reference

For the setting of these parameters, see "14-7-3. Specifying a Parameters for Torque-Speed Curve / Efficiency Map" on page 1199.

14-5-3. Map View Window

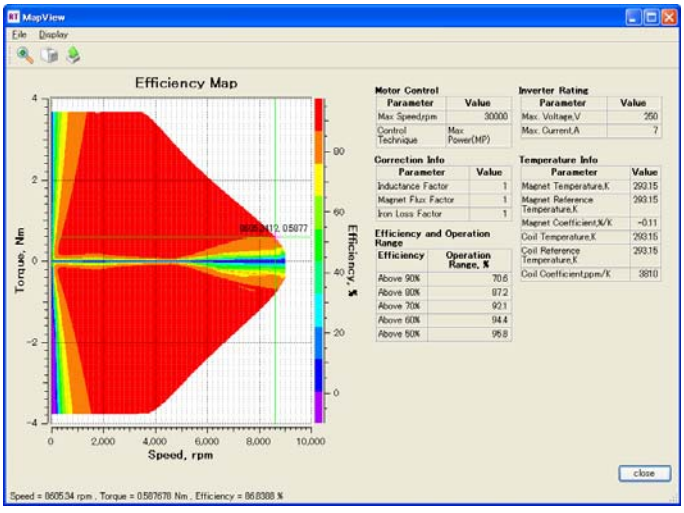
The map view window opens when the [Torque-speed curve] button or the [Efficiency map] button is clicked to display the graph.

The left side of the screen is the torque-speed curve / efficiency map and on the right side is the parameter for the torque-speed curve / efficiency map. The percentage of operating range is displayed to show how much of the efficiency is acquired in the entire operating range of the efficiency map.

After displaying the torque-speed curve in this window, the graph can be changed to the d/q current-speed curve (N-Id/Iq) or to the current amplitude-speed curve (N-Iam). Also, it is possible to change the efficiency map to the iron loss map (N-T-Iron loss) and to the coil loss map (N-T-Coil loss).

To zoom in on a part of the graph, click the [Zoom] tool button and then click and drag on the graph window the region to be zoomed. The reset the zoom click the right mouse button. To exit the zoom mode and to reset the zoom, click the [Zoom] tool button again.

When the mouse cursor is over the torque-speed curve, the rotation speed and the torque at the point can be displayed at the bottom of the window. In a map view, the value of the displayed quantity, for example efficiency, will also be shown.



Map view window (Efficiency map)

Reference

For display items, see "14-7-7. Displaying the Type of the Torque-Speed Curve / Efficiency Map" on page 1204.

14-6. Menu Bar

14-6-1. Menu Bar of the Main Window

The following functions are available from the menu.

■ [File]

- [Open]
  This menu item is used to open the JMAG-RT model data file (*.rtt).
- [Open Setting]
  This menu item is used to load previously saved settings for the detailed parameters saved in an XML file(*.xml).
- [Save Setting]
  This menu item is used to save the settings for the detailed settings window to an XML file.
- [Exit]
  This menu item terminates the JMAG-RT Viewer.

■ [Tools]

- [Torque-Speed curve]
  Selecting this menu item will display the torque-speed curve with the settings in the detailed setting option window. This is the same as clicking the [Torque-Speed curve] button .
- [Efficiency map]
  Selecting this menu item will display the efficiency map with the settings in the detailed setting option window. This is the same as clicking the [Efficiency map] button .
- [Language]
  Selecting this menu item will change the display language. English and Japanese are supported.
- [Temperature Unit]
  This menu selects the unit of temperature used for the temperature correction of the coil and magnet in the detailed setting option window. The options are Kelvin (K), Celsius (deg C) and Fahrenheit (deg F).

  MEMO When the [Temperature Unit] is changed after a value has been input in the [Temperature] text box, [Base Temperature] text box or [Coefficient Temperature], the value is automatically converted to the value in the specified unit.

■ [Help]

- [Help]
  Selecting this menu item will start the online help for the JMAG-RT Viewer.
- [Registration Key Update]
  Selecting this menu item allows the setting of the license key for displaying the torque-speed curve. The license key can be requested from the JSOL web site. When this menu item is selected, the [Registration Key Setup] dialog opens. Enter your [E-mail address], [Expiry date] and [License Key] in the dialog to set the license.
- [About]
  Selecting this menu will display the version information of the JMAG-RT Viewer.

14-6-2. Menu Bar of the Map View Window

The following functions are available from the menu.

■ [File]

- [Export]
  Selecting this menu item will export the torque-speed curve or the efficiency map to a report (*.pdf) or to an image file (*.bmp, *.jpeg, *png, *.tif). The [Export] icon in the tool bar can also be used to export the image.

  ▶▶ Reference
  For details, see "14-7-8. Exporting a Report of Torque-Speed Curve/Efficiency Map" on page 1205.
- [Csv Export]
  Selecting this menu item exports the data of the torque-speed curve or the efficiency map as .csv format file.

  ▶▶ Reference
  For details, see "14-7-9. Exporting the Tabular Data of Torque-Speed Curve / Efficiency Map" on page 1206.

- [Print Setup]
  This menu sets the properties for printing reports and images.
- [Print Preview]
  Selecting this menu item will display a preview of the page for printing.
- [Print]
  Selecting this menu will print a report of the torque-speed curve or the efficiency map to the selected printer. The function is the same as [Print] icon in the tool bar.
- [Close]
  Close the map view window.

■ [Display]

- [Display]
  This menu allows the type of graph displayed in the window to be changed.

  ▶▶ Reference
  For details, see "14-7-7. Displaying the Type of the Torque-Speed Curve / Efficiency Map" on page 1204.
- [Set Color Range]
  Select this menu item to display a dialog where the range and color parameters of the graph can be changed.
  - [Min. Value] / [Max. Value]
    Set the range of the color bar.
  - [Gradient]
    Select the color gradient that will be used for the graph. Available options are [Purple-Red], [Blue-Red], [Gray-scale] and [CMR]

  MEMO [CMR] is optimized for printing.

14-7. Running Procedures

14-7-1. Importing a JMAG-RT Model Data File

JMAG-RT model data file (*.rtt) can be opened using either the menu or by dropping the file onto the JMAG-RT Viewer.

▶▶ Reference

For the notes, see "14-3. Notes Regarding the JMAG-RT Model Data File" on page 1190.

■ Operation from the menu bar.

1. **Select [File] > [Open] from the menu bar.**

   [Open] dialog is opened.

2. **Select the folder where the file to open is located.**

3. **Select the file or put the file name in the file name box.**

4. **Click [OK].**

   The file is opened and the basic information of the JMAG-RT model is displayed in the main window.

■ Operation by drag and drop

1. **Dragging the JMAG-RT model data file(*.rtt) and drop on the main windows of JMAG-RT Viewer.**

   The file is imported and the basic information of the JMAG-RT model is displayed in the main window.

14-7-2. Displaying the Ld/Lq Map

1. **After opening the JMAG-RT model data file, click [Ld map] or [Ld map] in the main window.**

   [Ld/Lq map] is opened with the map.

   MEMO To zoom in on part of the Ld map and Lq map, click the mouse and drag a rectangle for the region to zoom after the [Zoom] icon is clicked.

   MEMO When the mouse cursor is over the Ld-Lq map the rotation speed, the torque and the inductance at the point can be displayed in the bottom of the window.

14-7-3. Specifying a Parameters for Torque-Speed Curve / Efficiency Map

1. Click [More] on the main window.

The window expands to show the detailed setting option window on the right side.

2. The following parameters can be specified.

MEMO The short title in brackets shown with the parameter, i.e. Vmax with Max. Voltage (Vmax), is used as the label in the report(*.pdf) of the torque-speed curve / efficiency map.

MEMO Parameters can be exported to an XML file.

▶▶ Reference

- For exporting the report of the torque-speed curve / the efficiency map, see "14-7-8. Exporting a Report of Torque-Speed Curve/Efficiency Map" on page 1205.
- For exporting parameters to the file, see "14-7-4. Saving the Settings for Parameters for Torque-Speed Curve / Efficiency Map to a File" on page 1202.

■ [Inverter Rating]

The specification of the inverter.

- [Max. Voltage (Vmax)]
  The bus voltage of the inverter. The unit is volt.
- [Max. Current (Imax)]
  The maximum current of the inverter. The unit is ampere.

■ [Motor Control]

Specifies the motor control information.

- [Control Type]
  The following are the types of control methods.
  - [id=0 (id=0)]
  - [Max. Torque/Current (MTPA)]
  - [Max. Torque/Flux (MTPF)]
  - [Max. Efficiency (ME)]
  - [Max. Power (MP)]
  - [Moment of Force (PF=1)]
  - [Flux Weakening (FW)]
- [Max. Speed [rpm]]

MEMO Multiple control methods cannot be combined and used.

▶▶ Reference
For more information about the important notes regarding the motor control information, "14-8. Motor Control Information" on page 1207.

■ [Calculation Resolution]

To specify a calculation resolution, click to check on [Calculation Resolution] to set following:

- [Current Divisions (Idiv)]
- [Phase Step (βdiv)]
  The unit is degree.
- [Speed Interval (Ndiv)]
  The unit is rpm.

MEMO Finer calculation resolutions improve the accuracy of the torque-speed curve and the efficiency map but increase the calculation time.

■ [Correction Coefficients]

The following correction coefficients can be specified if required.

- [Coefficient of Inductance (Lcof)]
- [Coefficient of Iron Loss (Hcof)]
- [Coefficient of the Magnet Flux (ϕcof)]

Important **If the iron loss is not considered when creating the JMAG-RT model, the effect is not included in the calculation of the Torque-Speed curve / Efficiency map even [Coefficient of Iron Loss] if an Iron Loss correction is specified.**

■ [Coil Temperature Correction], [Magnet Temperature Correction]

To apply a correction for the coil temperature, check [Coil Temperature Correction] and input [Temperature], [Base Temperature] and [Coefficient Temperature]. To apply a correction for the magnet temperature, check [Magnet

Temperature Correction] and input parameters. To input these parameters, see the following correction equations.

Correction equation for the magnet temperature:

$$F' = F\left(1 + \frac{k}{100}(Tf_{now} - Tf_{base})\right)$$

F': magnetic flux after the correction (Wb)

F: magnetic flux before the correction (Wb)

k: correction coefficient of magnet temperature (%/K)

* range: - 100.0 - 0.0 (negative value)

default: -0.11%/K

Tf_{now}: Magnet temperature (K)

Tf_{base}: Base Temperature (K)

* default value: 293.15 K

Correction equation for the coil temperature:

$$R' = R(1 + \alpha(Tr_{now} - Tr_{base}))$$

R': coil resistance after the correction

R: coil resistance before the correction

α: correction coefficient of coil temperature (ppm / K)

* range: > = 0.0

default: 3810 ppm/K

Tr_{now}: Coil Temperature (K)

Tr_{base}: Base Temperature (K)

* default: 293.15 K

14-7-4. Saving the Settings for Parameters for Torque-Speed Curve / Efficiency Map to a File

The parameters set in the detailed settings can be exported to a file (*.xml).

1. **Select [File] > [Save Setting] from the menu bar in the Map view window.**
   [Save xml file] dialog is opened.
2. **Specify the file name to save in the [File name] text box.**
3. **Specify the location to save the file.**
4. **Click [save] to save the file.**
   The setting file is exported in the specified folder.

14-7-5. Importing the Setting File of Parameters for Torque-Speed Curve / Efficiency Map

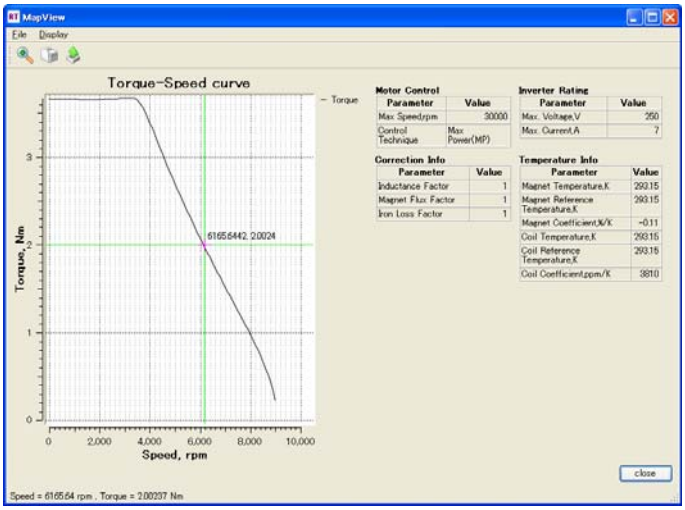
The parameters for the motor control settings can be imported from a previously saved file (*.xml).

1. **Select [File] > [Open Setting] from the menu bar in the Map view window.**
   [Open setting file] dialog is opened.
2. **Select the folder where the setting file to import is saved.**
3. **Select the file or put the file name in the file name box.**
4. **Click [Open].**
   The setting file is imported and parameters in the detailed setting option window are updated.

14-7-6. Displaying a Torque-Speed Curve / Efficiency Map

1. **Click [Torque-Speed curve] or [Efficiency map] in the detailed setting option window, or select [Tool] > [Torque-Speed curve] or [Efficiency map] from the menu bar.**

   The map view window is opened.



Map view window (Torque-speed curve)

14-7-7. Displaying the Type of the Torque-Speed Curve / Efficiency Map

After drawing the torque-speed curve in this window, the diagram can be changed to the d/q current-speed curve (N-Id/Iq) and to the current amplitude-speed curve (N-Iam).

Also, it is possible to change the efficiency map to the iron loss map (N-T-Iron loss) and to the coil loss map (N-T-Coil loss).

1. Click [Torque-Speed curve] or [Efficiency map] in the detailed setting option window.

The map view window is opened.

2. Click [Display] on the menu bar and select the type of the diagram.

For the torque-speed curve display, the diagram can be switched to the following types.

- N-T (Torque-Speed curve)
- N-Id/Iq (d/q Current-Speed curve)
- N-Iam (Current Amplitude-Speed curve)

For the display window of the efficiency map, the diagram can be changed to the following types.

- N-T-Efficiency (Efficiency map)
- N-T-IronLoss (Iron loss map)
- N-T-CoilLoss (Coil loss map)
- N-T-Coil/IronLoss (Coil loss / Iron loss map)
- N-T-Id (d axis current map)
- N-T-Iq (q axis current map)
- N-T-Iam (Phase current amplitude map)
- N-T-ILd (d axis current / inductance map)
- N-T-ILq (q axis current / inductance map)

MEMO Notations are as followings.
- N: Rotation speed (rpm)
- T: Torque (Nm)
- Id: d axis amplitude (A)
- Iq: q axis amplitude (A)
- Iam: phase current amplitude (A)
- Ld: d axis inductance (H)
- Lq: q axis inductance (H)

Important **The value on the N-T-Id/Iq or N-T-Id/Iq in the map view window is Ld/Lq or Id/Iq. It is different from the value which is shown when [Ld map] or [Lq map] is displayed from the main window.**

14-7-8. Exporting a Report of Torque-Speed Curve/Efficiency Map

A report for the torque-speed curve or the efficiency map can be exported to a pdf file or an image file.

1. Select [File] > [Export] from the menu bar in the map view window.

[Export File Name] dialog box is displayed.

2. Select the file type from the [Save as Type] list.

The type of the files that can be selected are as follows:

- PDF file (*.pdf)
- SVG file (*.svg)
- PostScript file (*.ps)
- Image file (*.bmp, *.ico, *.jpeg, *.png,*.ppm, *.tif, *.xbm, *.xpm)

3. Enter a file name in the [File name] text box.

When the [Image file] is selected as the file type in step.2, specify the file name with the extension of the image file type (i.e. motor.png).

4. Specify the directory to save the file.

5. Click [Save].

The report is saved in the specified folder.

MEMO In the report file, the parameters used to calculate the graph are also listed in addition to the diagram. Labels in the list corresponded to the short title on the detailed setting option window.

▶▶ Reference

For parameters in the detailed setting option window, see "14-7-3. Specifying a Parameters for Torque-Speed Curve / Efficiency Map" on page 1199.

14-7-9. Exporting the Tabular Data of Torque-Speed Curve / Efficiency Map

Tabular numeric data for the torque-speed curve or the efficiency map can be exported to a csv file This can then be opened in a spreadsheet such as Microsoft Excel.

■ Export data of Torque-Speed curve

1. Select [File] > [Csv Export] from the menu bar.

[Save csv file] dialog box is displayed.

2. Enter a file name in the [File name] text box.

3. Specify the directory to save the file.

4. Click [Save].

The file is saved in the specified folder.

■ Export data of Efficiency map

The efficiency map data is exported as values on a grid to a .csv file.

1. Select [File] > [Csv Export] from the menu bar.

[Csv Export] dialog box is displayed.

2. Set [Grid Resolution].

For both the [Horizontal Axis] and [Vertical Axis], set the following parameter to specify the grid resolution.

- [Min. Value]
- [Max. Value]
- [Divisions]

3. Click the button on the right side of the [CSV File] text box.

[Save csv file] dialog box is displayed.

4. Enter a file name in the [File name] text box.

5. Specify the directory to save the file.

6. Click [Save].

[Save csv file] dialog box is closed to back to [Csv Export] dialog.

7. Click [Save].

The file is saved in the specified folder.

14-8. Motor Control Information

14-8-1. Algorithm for the Calculation of the Torque-Speed Curve

- The speed range of the motor is split into several ranges.
- The motor control proceeds from the control method. Using this control technique the calculation is performed to obtain the current vector, inductance (L_d, L_q) and magnetic flux Ψ_a, then the torque.
- If one of the constraints for the control technique is exceeded, then the next control method defined in the list is used.

Important **When [Max. Efficiency] is selected for [technique], the current vector is defined as to maximize the efficiency, so the torque curve can be significantly changed.**

MEMO The JMAG-RT model data file (*.rtt) used to draw torque-speed curves in JMAG-RT Viewer uses the [Ld, Lq model] information that is specified with the [Spatial Harmonic Model] information.
The model information used from the JMAG-RT model data file (*.rtt) can be confirmed in the [Type] column in [Motor Info] of the main window in JMAG-RT Viewer.

Reference
For more information about the main window in JMAG-RT Viewer, see "14-5-1. Main Window of JMAG-RT Viewer" on page 1192.

14-8-2. Algorithm for the Calculation of the Efficiency Map

- When calculating the efficiency map, the Torque-Speed curve is first obtained.Then the maximum efficiency at each point below the curve is found.
- Starting from the current which maximizes the torque, an analysis is performed to evaluate the torque, the coil loss and the iron loss, over a range of current amplitudes and phases.

14-8-3. Important Notes Regarding Motor Control and Limitations of Motor Control

This section explains the motor control techniques and their limitations.

In this section, following notations are used.

L_d : d axis inductance (H)

L_q : q axis inductance (H)

Ψ_a : magnetic flux with no load (effective line flux) (Wb)

R_a : phase resistance (ohm)

I_d : d axis current (A)

I_q : q axis current (A)

I_a : effective current $(=\sqrt{i_d^2 + i_q^2})$

V_{am} : limit voltage (V)

14-8-3-1. id=0

For the specified current, the d axis current i_d to is controlled so that it is 0. This means that the current phase is 0 degrees. In this case, i_d and i_q are expressed as eq. (1).:

$$\begin{cases} i_d = 0 & \\ i_q = I_a & \text{when operating as a motor} \\ i_q = -I_a & \text{when operating as a generator} \end{cases} \quad \cdots\cdots\cdots\cdots (1)$$

14-8-3-2. Max. Torque / Current

For the specified current, the current phase is controlled so as to maximize the torque.

The current phase β to be controlled is expressed as in eq.(2).

$$\sin\beta = \frac{-\Psi_a I_a \pm \sqrt{\Psi_a^2 I_a^2 + 8(L_q - L_d)^2 I_a^4}}{4(L_q - L_d) I_a^2} \quad \cdots\cdots\cdots\cdots\cdots\cdots (2)$$

Note that β is in the range $(90 \geq \beta \geq -90)$ when operating as a motor, and in the range $(270 \geq \beta \geq 90)$ when operating as a generator.

In the case where $L_d = L_q$, this technique is identical with the “id=0” control technique.

14-8-3-3. Max. Torque / Flux

This control method selects among several conditions which generate the same torque, for the one which minimizes the linkage flux. i_d and i_q are as expressed as in eq.(3).

$$\begin{cases} i_d = -\dfrac{\Psi_a + \Delta\Psi_d}{L_d} & \\ i_q = \dfrac{\sqrt{\Psi_0^2 - \Delta\Psi_d^2}}{L_q} & \text{when operating as a motor} \\ i_q = -\dfrac{\sqrt{\Psi_0^2 - \Delta\Psi_d^2}}{L_q} & \text{when operating as a generator} \end{cases} \quad \cdots\cdots (3)$$

where,

$$\Delta\Psi_d = \frac{-L_q\Psi_a \pm \sqrt{L_q^2\Psi_a^2 + 8(L_d - L_q)^2\Psi_0^2}}{4(L_q - L_d)} \quad \cdots\cdots (4)$$

$$\Psi_0 = \sqrt{(L_d i_d + \Psi_a)^2 + (L_q i_q)^2} \quad \cdots\cdots (5)$$

and when $L_d = L_q$, these expressions derives eq.(6).

$$\begin{cases} i_d = -\dfrac{\Psi_a}{L_d} & \\ i_q = \dfrac{\Psi_0}{L_q} & \text{when operating as a motor} \\ i_q = -\dfrac{\Psi_a}{L_q} & \text{when operating as a generator} \end{cases} \quad \cdots\cdots (6)$$

14-8-3-4. Flux Weakening

The flux Weakening control varies the phase of the current for a constant current amplitude so that the voltage is lower than the limit voltage for a specified current. i_d and i_q are as in e.q. (7).

$$\begin{cases} i_d = \dfrac{L_d\Psi_a \pm \sqrt{L_q^2\Psi_a^2 + (L_q^2 - L_d^2)\left(L_q^2 I_a^2 - \left(\dfrac{V_{om}}{\omega}\right)^2\right)}}{(L_q^2 - L_d^2)} \\ i_q = \sqrt{I_a^2 - i_d^2} \quad \text{when operating as a motor} \\ i_q = \sqrt{I_a^2 - i_d^2} \quad \text{when operating as a generator} \end{cases} \quad \cdots\cdots(7)$$

where

$$V_{om} = \frac{V_{am}}{\sqrt{2}} - R_a I_a \quad \cdots\cdots(8)$$

When $L_d = L_q$, these expressions derives eq.(9).

$$\begin{cases} i_d = \dfrac{\left(\dfrac{V_{om}}{\omega}\right)^2 - \Psi_a^2 - L_q^2 I_a^2}{2L_d\Psi_a} \\ i_q = \sqrt{I_a^2 - i_d^2} \quad \text{when operating as a motor} \\ i_q = \sqrt{I_a^2 - i_d^2} \quad \text{when operating as a generator} \end{cases} \quad \cdots\cdots(9)$$

14-8-3-5. Moment of Force 1

For a given current, the current phase is controlled so that the moment of force is 1. i_d and i_q are expressed as e.q.(10).

$$\begin{cases} i_d = \dfrac{\Psi_a - \sqrt{\Psi_a^2 + 4(L_q - L_d)L_q I_a^2}}{2(L_q - L_d)} & \\ i_q = \sqrt{I_a^2 - i_d^2} & \text{when operating as a motor} \\ i_q = -\sqrt{I_a^2 - i_d^2} & \text{when operating as a generator} \end{cases} \quad \cdots\cdots (10)$$

When $L_d = L_q$, eq.(11) is derived.

$$\begin{cases} i_d = \dfrac{-L_q I_a^2}{\Psi_a} & \\ i_q = \sqrt{I_a^2 - i_d^2} & \text{when operating as a motor} \\ i_q = -\sqrt{I_a^2 - i_d^2} & \text{when operating as a generator} \end{cases} \quad \cdots (11)$$

14-8-3-6. Max. Efficiency

For the specified current, the current phase is controlled so that the efficiency is maximized.

14-8-3-7. Max. Power

Searches for the maximum point in the torque while modifying the current amplitude and current phase.

14-9. Executing from the Command Line

The JMAG-RT Viewer can also be controlled from the command line.

Either to display the JMAG-RT Viewer window with rtt file and settings already loaded or to create a report or image file directly without displaying the GUI.

1. Start a command prompt and input the commands to run the viewer as described below.

Alternatively a batch file (*.bat) can be created the commands.

The syntax is as follows.

■ Syntax

```
# > "path to the rtviewer.exe" "path to the rtt file" [--
property name = value]
```

Example:

```
#> rtviewer.exe new_mtr600_Iron.rtt --in="sample.xml" --out="./
tests/exportwh500.png" --width=500 --height=500
```

MEMO To create a batch file, save the command above as a text file. Then change the extension to *.bat and then double click it to run.

Important **To set the command, the line should be set in a line. To continue to the next line, specify "^(hat)" at the end of the line.**

The properties that can be set from the command line are as follows.

property name (propName)	Type (Type)	Flag name (scriptKeyName)	Description
*rttfile	String		RTT file (without options)
in	String		Setting file (*.xml) This file is explained in the export procedures in P.1202 "14-7-4. Saving the Settings for Parameters for Torque-Speed Curve / Efficiency Map to a File". The [More] button in the main window of JMAG-RT Viewer is selected when a setting file is specified.
out	String		Export file

property name (propName)	Type (Type)	Flag name (scriptKeyName)	Description
subout	String		The export file for [Efficiency and Operation Range] that is displayed in map view. The file can be exported when the type is "Map" (EfficiencyMap or IronLossMap).
type	Flag		Type of the torque-speed curve and the efficiency map
		TorqueSpeedCurve	N-T (Torque-Speed curve)
		IdIqSpeedCurve	N-Id/Iq (d/q axis current - speed curve)
		IamSpeedCurve	N-Iam (phase current amplitude-speed curve)
		EfficiencyMap	N-T-Efficiency (Efficiency map)
		IronLossMap	N-T-IronLoss (Iron loss map)
		CopperLossMap	N-T-CoilLoss (Coil loss map)
		IronCopperLoss-Map	N-T-Coil/IronLoss (Iron loss / Coil loss map)
		IdMap	N-T-Id (d axis current map)
		IqMap	N-T-Iq (q axis current map)
		IamMap	N-T-Iam (phase current amplitude map)
		LdMap	N-T-ILd (d axis current / inductance map)
		LqMap	N-T-ILq (q axis current / inductance map)
width	Double		Width of image Valid for export Torque-Speed curve / Efficiency map to image file.
height	Double		Height of image Valid for export Torque-Speed curve / Efficiency map to image file.
haxis_min	Double		Minimum of horizontal axis for grid resolution Valid for export Efficiency map to csv file. If not specified, set automatically.
haxis_max	Double		Maximum of horizontal axis for grid resolution Valid for export Efficiency map to csv file. If not specified, set automatically.

property name (propName)	Type (Type)	Flag name (scriptKeyName)	Description
haxis_div	Double		Number of division of horizontal axis for grid resolution Valid for export Efficiency map to csv file. If not specified, set 10 as default.
vaxis_min	Double		Minimum of vertical axis for grid resolution Valid for export Efficiency map to csv file. If not specified, set automatically.
vaxis_max	Double		Maximum of vertical axis for grid resolution Valid for export Efficiency map to csv file. If not specified, set automatically.
vaxis_div	Double		Number of division of vertical axis for grid resolution Valid for export Efficiency map to csv file. If not specified, set 10 as default.
language	Flag	 ja en	Language If not specified, the setting in the registry is used. Japanese English
color_min	Double		Minimum of the color range
color_max	Double		Maximum of the color range If not specified, set automatically.
gradient_type	Flag	 purple_red blue_red gray_scale cmr	Type of color gradation purple - red blue - red gray CMR
papersize	Flag	 A4 Letter	Size of print Valid for printing Torque-Speed curve or Efficiency map to PDF or PS A4 Letter

14-9-1. Setting the License Check (License Waiting Function)

The function ends with an error if no license is temporarily available, such as when the license is used by other users and the function is used.

The calculation can be suspended until the license becomes available by changing the environment variables by setting [JMAG_LICENSE_CHECK_WAITING] in JMAG-RT Viewer.

MEMO The process is in "running" sequence while waiting to use a license.

■ Important Notes Regarding the Use of This Function

- The [License Waiting Function] is valid when [--out] is included in the command as an argument running from the command line.

  Example: A command that has the argument [--out] included.

  ```
  \>rtviewer.exe motor.rtt --in="sample.xml" --out="./jmag/N-T-Eff.png"
  ```

- The [License Waiting] function is invalid when operating from the GUI (operating from the graphics window).

■ Setting JMAG_LICENSE_CHECK_WAITING (Windows)

The procedure to set the environment variable for Windows XP is as follows.

1. **Double-click [Control Panel].**
2. **Double-click [System].**
3. **Click the [Advanced] tab, and then click [Environment Variables].**
4. **Click [New] under [System variables] to open the [New System Variable] dialog box.**
5. **Enter "JMAG_LICENSE_CHECK_WAITING" in the [Variable name] text box and "1" in the [Variable value] text box.**

   MEMO If "0" is entered in the [Variable value] text box, the calculation ends with an error if there is no license available.

6. **Click [OK] to close all the dialog boxes.**

14-10. Error / Warning Codes

The list of the errors and warnings that may occur when performing a calculation for the torque speed curve or efficiency map.

■ Error Codes

Code ID	Description	
1	Title	Speed Interval is too large.
	Comment	A smaller interval for the speed should be set
2	Title	Number of current divisions is 1 or smaller.
	Comment	The number of current divisions should be greater than 1.
3	Title	Failed to open RTT file.
	Comment	This filename might be incorrect. Please check the file is a valid JMAG-RT rtt file.
4	Title	Failed to display Torque-Speed curve / Efficiency map because this RTT file is invalid.
	Comment	The Torque-Speed curve / Efficiency map can only be displayed if both of the following criteria are met: • Three-phase PM Synchronous Motor should be selected.
5	Title	Phase Step value is too large.
	Comment	Set a smaller value for the phase step.
6	Title	Max. Voltage is less than 0.
	Comment	The maximum voltage should be set to a value greater than zero.
7	Title	Max. Current is less than zero.
	Comment	The maximum current should be set to a value greater than zero.
8	Title	Speed interval is less than 0.
	Comment	The speed interval should be set to a value that is greater than zero.
9	Title	Phase Step is less than 0.
	Comment	The phase step should be set to a value greater than zero.
10	Title	Current Divisions is less than 0.
	Comment	The current divisions should be set to a value greater than zero.
11	Title	Torque-Speed curve / Efficiency map cannot be displayed because there is no inductance information in the RTT file.
	Comment	To display the torque-speed curve or efficiency map the Ld-Lq inductance information is required.

Code ID	Description	
12	Title	Torque-Speed curve / Efficiency map cannot be displayed because the inductance information in the RTT file is incorrect.
	Comment	The resolution of inductance is set to zero. This file cannot be used to display the Torque-Speed curve / Efficiency map
15	Title	Calculation is cancelled.
	Comment	The diagram is not displayed since the calculation was cancelled. Retry the calculation.
16	Title	Failed to calculate under the specified motor control information.
	Comment	Try followings • Increase the Max. Voltage. • Increase the Max. Current. • Change motor control technique.
17	Title	Coil resistance becomes negative.
	Comment	Check the temperature correction parameters. The base temperature and the coefficient temperature should be set so as not to make the corrected coil resistance negative.
18	Title	Magnet flux becomes negative.
	Comment	Check the temperature correction parameters. The base temperature and the coefficient temperature should be set so as not to make the corrected magnet flux negative.
19	Title	Insufficient memory to calculate with required resolution.
	Comment	The analysis did not have enough memory to complete since the required resolution is too fine. To reduce the resolution, try the following. • Reduce the current divisions. • Increase the phase step • Increase the speed interval Note that the upper resolution depends on the amount of available memory.

Code ID	Description	
20	Title	Failed to calculate with “Max. Torque/Flux” technique.
	Comment	The “Max. Torque/Flux” technique cannot be used for low rotation speeds. Change the technique or to control using the “Max. Torque/Flux” technique, try followings. • Add another control method to the table which is used for the low rotation speeds. • Increase the Max. Voltage • Increase the Max. Current
21	Title	Failed to calculate with the “Flux Weakening” technique.
	Comment	The “Flux Weakening” technique cannot be used at low rotation speeds. Change the technique or to control using the “Flux Weakening” technique, try following. • Add another control method to the table which is used for the low rotation speeds. • Increase the Max. Voltage • Increase the Max. Current
22	Title	Failed to calculate with “Moment of Force 1” technique.
	Comment	To control by “Moment of Force 1” technique, try followings. • Reduce the Max. Current. • Increase the Max. Voltage.
23	Title	Max. Speed is less than 0.
	Comment	The maximum speed should be set to a value greater than zero.
24	Title	Max. Speed is too small.
	Comment	A larger maximum speed should be set.

■ Warning Codes

Code ID	Description	
101	Title	Out of range calculation for the data in the RTT file. The Max. Current is too large.
	Comment	The current data used in the calculation exceeds the range of the values in the inductance table or the iron loss table in the RTT file. The calculation might not be performed correctly for such cases. This warning can be fixed by following: • Reduce the Max. Current • Use an RTT file which is defined for a greater than the Max. Current.

Code ID	Description	
102	Title	Out of range calculation for the data in the RTT file. The accuracy of the calculation may be reduced.
	Comment	The phase data used in the calculation exceeds the range of values in the inductance table or the iron loss table in the RTT file. The calculation might not be performed properly for such cases. This warning can be fixed by following. • Use an RTT file which is defined for the whole 360 degrees in the current phase.
103	Title	Out of range calculation for the data in the RTT file. The Max. Speed is too high.
	Comment	The rotation speed data used in the calculation exceeds the range of values in the inductance table or the iron loss table in the RTT file. The calculation might not be performed properly for such cases. This warning can be fixed by following. • Reduce the Max. Speed. • Use an RTT file which is defined with a rotation speed greater than the Max. Speed.
104	Title	Some part of the calculation did not converge. The accuracy of the calculation may be reduced.
	Comment	The convergence of calculation to account for the nonlinearity of inductance did not converge at some calculation points. The accuracy of the current vector or inductance used to derive the torque and the efficiency may be reduced.
105	Title	No iron loss information in the RTT file. The displayed Torque-Speed curve / Efficiency map does include the effect of the iron loss.
	Comment	To save the iron loss information to the RTT file, select the option to take the iron loss into account when creating the model using JMAG-RT.
106	Title	The Displayed Torque-Speed curve/Efficiency map does not take the iron loss into account, because the iron loss information of the RTT file is invalid.
	Comment	The iron loss data in the RTT file is incorrect and cannot be used for the calculation.

Chapter 15. Editing a JCF File (Simple FHI)

This chapter describes how to edit a JCF file.

▶▶ Reference

For information about each parameter in a JCF file, see the PDF file in the JMAG install folder/documents.

15-1. Overview

Unlike solver files (text format), JCF files are stored in binary format. A file access library, SimpleFHI, is required to access to a JCF file.

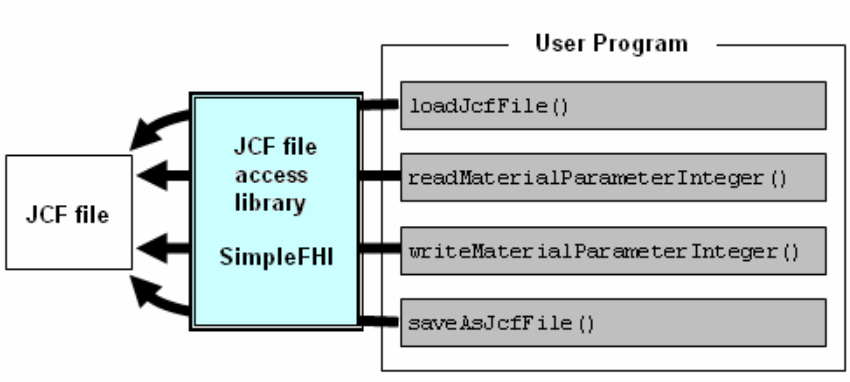
An external program can confirm and edit the values specified for parameters using SimpleFHI, and save the changes in a new file.

■ Editing a JCF file

1. Write program codes appropriate for changing parameters.

2. Link the program to the file access library, SimpleFHI.

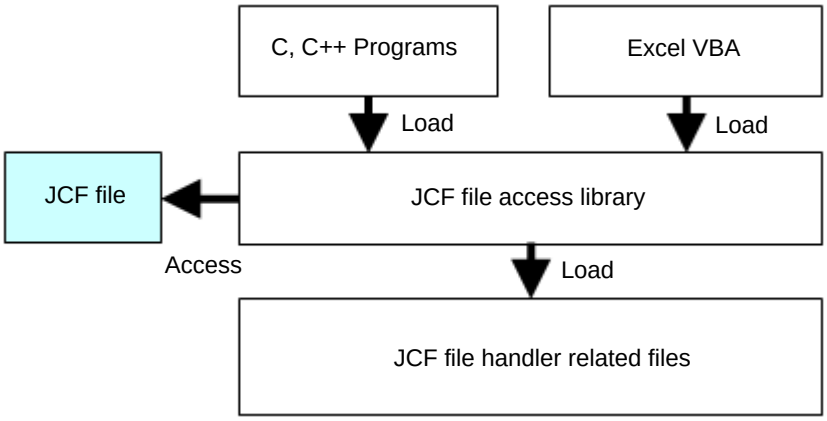
3. Build the program, and then execute it.



Editing a JCF file

■ Access to a JCF file

A JCF file can be accessed as follows.



Access to a JCF file

The library of SimpleFHI loads some JCF file handlers as sub- libraries.

15-2. Environment Required for Editing JCF Files

15-2-1. Programs

Either one of the following programs is required.

- Excel (Windows only)
- C, C++ programs (Windows/Linux)

15-2-2. Files

The following files are required to access a JCF file.

	Windows	Linux
JCF file access library	• SimpleFHI_32_10.dll (INS_DIR)	N/A (The static library is used.)
Files required to build the C and C++ programs	• SimpleFHI.h • SimpleFHI.lib (INS_DIR\Tools\SimpleFHI)	• libSimpleFHI.a (INS_DIR/solver/mod/tools/SimpleFHI/lib) • SimpleFHI.h (INS_DIR/solver/mod/tools/SimpleFHI/include)
JCF file handler files (required to build and execute a program)	• filehandler.dll • icudt20.dll • icuuc20.dll • xerces-c_1_6_0.dll (INS_DIR)	• libFileHandler.a • libicudtata.so • libicuuc.so • libxerces-c1_6_0.so For 32bit (INS_DIR/solver/mod/lib/linux32) For 64bit (INS_DIR/solver/mod/lib/linux64)

Important **INS_DIR indicates the JMAG installation directory.**
e.g. JMAG-Designer: C:\Program Files\JMAG-Designer12.0
e.g. Linux: /usr/works

The following files are provided as samples.

■ Sample code (C, C++ programs)

■ Location (Windows):

```
\(JMAG installation directory)\Tools\SimpleFHI\samples\C
```

- main.c
  This file contains sample source codes.
- sample.dsp
  This file is a project file containing sample source codes.
- sample.dsw
  This file is a workspace file containing sample.dsp.
- sample.jcf
  This is a sample JCF file.

■ Location (Linux):

```
(installation directory)/solver/mod/tools/SimpleFHI/samples/
C/Sample
```

- main.c
  This file contains sample source codes.
- sample.jcf
  This is a sample JCF file.
- Makefile
  This file is needed for a build.
- SimpleFHI
  This is a program name created after a build.

■ Sample code (Excel VBA)

■ Location (Windows only):

```
\(JMAG installation directory)\Tools\SimpleFHI\samples
\ExcelVBA
```

- check_data.jcf
  This is a sample JCF file.
- declaration.txt
  This file contains declarations to reference functions provided by SimpleFHI. This file needs to be copied to a macro.
- sample.xls
  This file contains a sample Excel spread sheet and VBA source codes.

15-3. Functions of JCF File Access Library (SimpleFHI)

The JCF file access library, SimpleFHI, contains the following functions.

The terms [in] and [out] used in the following description indicate "input" and "output," respectively.

■ Functions for initialization

```
long loadJcfFile
(const char* path, JcfDataPtr* result);
```

Description	Opens a JCF file, and initializes the data for management.
Parameters	path[in] : path to an input JCF file result[out] : pointer-indicating data for management
Return value	1: success, 0: failure

■ Functions for handling files

```
long saveAsJcfFile
(JcfDataPtr data, const char* path);
```

Description	Attaches a name to a JCF file and stores the resulting file.
Parameters	data[in] : pointer-indicating data for management path[out] : path to a location where the JCF file is stored.
Return value	1: success, 0: failure

```
void closeJcfFile
(JcfDataPtr data);
```

Description	Closes the JCF file linked to the data indicated by the data pointer "data".
Parameters	data[in] : pointer indicating data for management
Return value	none

■ Functions for control parameters

```
long readControlParameterInteger
(JcfDataPtr data, const char* moduleName, const char*
paramName, long* result);
```

Description	Reads control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter result[out] : obtained value (integer)
Return value	1: success, 0: failure

```
long readControlParameterDouble
(JcfDataPtr data, const char* moduleName, const char*
paramName, double* result);
```

Description	Reads control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter result[out] : obtained value (floating point number)
Return value	1: success, 0: failure

```
long readControlParameterComplex
(JcfDataPtr data, const char* moduleName, const char*
paramName, double* real, double* img);
```

Description	Reads control parameters.
Parameters	data[in]pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter real[out] : obtained value (real part of a complex number) img[out] : obtained value (imaginary part of a complex
Return value	1: success, 0: failure

```
long readControlParameterString(JcfDataPtr data, const char*
moduleName, const char* paramName, char* buff, int n);
```

Description	Reads control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter buff[out] : buffer for storing a string obtained n[in] : size of buffer for the string
Return value	1: success, 0: failure

```
long writeControlParameterInteger
(JcfDataPtr data, const char* moduleName, const char*
paramName, long value);
```

Description	Writes control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter value[in] : written value (integer)
Return value	1: success, 0: failure

```
long writeControlParameterDouble
(JcfDataPtr data, const char* moduleName, const char*
paramName, double value);
```

Description	Writes control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter value[in] : written value (floating point number)
Return value	1: success, 0: failure

```
long writeControlParameterComplex
(JcfDataPtr data, const char* moduleName, const char*
paramName, double real, double img);
```

Description	Writes control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter real[in] : written value (real part of a complex number) img[in] : written value (imaginary part of a complex number)
Return value	1: success, 0: failure

```
long writeControlParameterString
(JcfDataPtr data, const char* moduleName, const char*
paramName, const char* str);
```

Description	Writes control parameters.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of module paramName[in] : name of parameter str[in] : written string
Return value	1: success, 0: failure

■ Functions for setting conditions

```
long createCondition
(JcfDataPtr data, const char* condName);
```

Description	Creates conditions.
Parameters	data[in] : pointer indicating data for management moduleName[in] : name of condition
Return value	1: success, 0: failure

```
void removeCondition
(JcfDataPtr data, const char* condName, int id);
```

Description	Deletes a condition by nominating the ID of the condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition
Return value	1: success, 0: failure

```
long countConditions
(JcfDataPtr data, const char* condName);
```

Description	Acquires the required number of conditions.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition
Return value	Number of conditions

```
long getIndexByConditionId
(JcfDataPtr data, const char* condName, int id);
```

Description	Acquires the ordinal number of a condition whose ID is specified.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition
Return value	Ordinal number
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readConditionParameterInteger
(JcfDataPtr data, const char* condName, int id, const char*
paramName, long* result);
```

Description	Reads condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter result[out] : obtained value (integer)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number ("id" is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long readConditionParameterDouble
(JcfDataPtr data, const char* condName, int id, const char*
paramName, double* result);
```

Description	Reads condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter result[out] : obtained value (floating point number)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number (“id” is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long readConditionParameterComplex
(JcfDataPtr data, const char* condName, int id, const char*
paramName, double* real, double* img);
```

Description	Reads condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter real[out] : obtained value (real part of a complex number) img[out] : obtained value (imaginary part of a complex number)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number (“id” is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long readConditionParameterString
(JcfDataPtr data, const char* condName, int id, const char*
paramName, char* buff, int n);
```

Description	Reads condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter buff[out] : buffer for storing a string obtained n[in] : size of buffer for the string
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number ("id" is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long readConditionParameterPoint
(JcfDataPtr data, const char* condName, int id, const char*
paramName, double* x, double* y, double* z);
```

Description	Reads condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter x[out] : obtained value (x-component) y[out] : obtained value (y-component) z[out] : obtained value (z-component)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number ("id" is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long writeConditionParameterInteger
(JcfDataPtr data, const char* condName, int id, const char*
paramName, long value);
```

Description	Writes condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter value[in] : written value (integer)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number (“id” is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long writeConditionParameterDouble
(JcfDataPtr data, const char* condName, int id, const char*
paramName, double value);
```

Description	Writes condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter value[in] : written value (floating point number)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number (“id” is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long writeConditionParameterComplex
(JcfDataPtr data, const char* condName, int id, const char*
paramName, double real, double img);
```

Description	Writes condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter real[in] : written value (real part of a complex number) img[in] : written value (imaginary part of a complex number)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number ("id" is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long writeConditionParameterString
(JcfDataPtr data, const char* condName, int id, const char*
paramName, const char* str);
```

Description	Writes condition parameters.
Parameters	data[in]pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter str[in] : written string
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number ("id" is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long writeConditionParameterPoint
(JcfDataPtr data, const char* condName, int id, const char*
paramName, double x, double y, double z);
```

Description	Writes condition parameters.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition id[in] : ID of condition paramName[in] : name of parameter x[in] : written value (x-component) y[in] : written value (y-component) z[in] : written value (z-component)
Return value	1: success, 0: failure
Remarks	For conditions without a conditional number (“id” is not included in the parameter names) such as the step condition, the conditional number is set to zero (0).

```
long countConditionGroups
(JcfDataPtr data, const char* condName, int id);
```

Description	Obtains the number of groups included in the specified condition.
Parameters	data[in]: pointer indicating data for management condName[in]: name of condition id[in]: ID of condition
Return value	Number of groups

```
long countConditionGroupElments
(JcfDataPtr data, const char* condName, int id, int
groupIndex);
```

Description	Obtains the number of elements in the group included in the specified condition.
Parameters	data[in]: pointer indicating data for management condName[in]: name of condition id[in]: ID of condition groupIndex[in]: Group ordinal number
Return value	Number of elements
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readConditionGroupType
(JcfDataPtr data, const char* condName, int id, int
groupIndex);
```

Description — Obtains the group type of the specified group by condition ID and group ordinal number. The following group types can be specified.
0 => Node
1 => Element
2 => Element face
3 => Element edge
4 => Region
5 => Region edge
6 => Solid
7 => Solid face

Parameters — data[in]: pointer indicating data for management
condName[in]: name of condition
id[in]: ID of condition
groupIndex[in]: Group ordinal number

Return value — Group type

Remarks — Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readConditionGroup
(JcfDataPtr data, const char* condName, int id, int groupIndex,
int elementIndex, long* firstId, long* secondId);
```

Description — Obtains the ID of the elements, nodes, and element faces in the specified group. A group can be specified by the condition ID or group ordinal number.

Parameters — data[in]: pointer indicating data for management
condName[in]: name of condition
id[in]: ID of condition
groupIndex[in]: Group ordinal number
elementIndex[in]: Group element ordinal number
firstId[out]: ID of an element or node
secondId[out]: ID of an element face

Return value — 1: success, 0: failure

Remarks — Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
void removeConditionByIndex
(JcfDataPtr data, const char* condName, int index);
```

Description	Deletes the condition specified by a ordinal number.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in]: Condition ordinal number
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readConditionParameterIntegerByIndex
(JcfDataPtr data, const char* condName, int index, const char*
paramName, long* result);
```

Description	Reads the parameters set for a condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in]: Condition ordinal number paramName[in]: Parameter name result[out]: Obtained value (integer)
Return value	1: success, 0: failure
Remarks	• For the conditions that do not have a condition ID (parameter name does not contain “id”) such as the Step condition, set Condition ID to zero (0). • Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readConditionParameterDoubleByIndex
(JcfDataPtr data, const char* condName, int index, const char*
paramName, double* result);
```

Description	Reads the parameters set for a condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in]: Condition ordinal number paramName[in]: Parameter name result[out]: Obtained value (float)
Return value	1: success, 0: failure
Remarks	• For the conditions that do not have a condition ID (parameter name does not contain "id") such as the Step condition, set Condition ID to zero (0). • Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readConditionParameterStringByIndex
(JcfDataPtr data, const char* condName, int index, const char*
paramName, char* buff, int n);
```

Description	Reads the parameters set for a condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in]: Condition ordinal number paramName[in]: Parameter name buff[out]: Buffer for storing the obtained string n[in]: Buffer size
Return value	1: success, 0: failure
Remarks	• For the conditions that do not have a condition ID (parameter name does not contain "id") such as the Step condition, set Condition ID to zero (0). • Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeConditionParameterIntegerByIndex
(JcfDataPtr data, const char* condName, int index, const char*
paramName, long value);
```

Description	Sets the parameters for a condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in] : Condition ordinal number paramName[in] : Parameter name value[in] : Output value (integer)
Return value	1: success, 0: failure
Remarks	• For the conditions that do not have a condition ID (parameter name does not contain “id”) such as the Step condition, set Condition ID to zero (0). • Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeConditionParameterDoubleByIndex
(JcfDataPtr data, const char* condName, int index, const char*
paramName, double value);
```

Description	Sets the parameters for a condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in] : Condition ordinal number paramName[in]: Parameter name value[in]: Output value (float)
Return value	1: success, 0: failure
Remarks	• For the conditions that do not have a condition ID (parameter name does not contain “id”) such as the Step condition, set Condition ID to zero (0). • Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeConditionParameterStringByIndex
(JcfDataPtr data, const char* condName, int index, const char*
paramName, const char* str);
```

Description	Sets the parameters for a condition.
Parameters	data[in] : pointer indicating data for management condName[in] : name of condition index[in] : Condition ordinal number paramName[in] : Parameter name str[in] : String to be output
Return value	1: success, 0: failure
Remarks	• For the conditions that do not have a condition ID (parameter name does not contain "id") such as the Step condition, set Condition ID to zero (0). • Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

■ Functions for setting materials

```
long countMaterials
(JcfDataPtr data);
```

Description	Acquires the number of materials whose name is specified.
Parameters	data[in] : pointer indicating data for management
Return value	Number of materials

```
long readMaterialParameterInteger
(JcfDataPtr data, int id, const char* paramName, long* result);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter result[out] : obtained value (integer)
Return value	1: success, 0: failure

```
long readMaterialParameterDouble
(JcfDataPtr data, int id, const char* paramName, double*
result);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter result[out] : obtained value (integer)
Return value	1: success, 0: failure

```
long readMaterialParameterComplex
(JcfDataPtr data, int id, const char* paramName, double* real,
double* img);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter real[out] : obtained value (real part of a complex number) img[out] : obtained value (imaginary part of a complex number)
Return value	1: success, 0: failure

```
long readMaterialParameterString
(JcfDataPtr data, int id, const char* paramName, char* buff,
int n);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter buff[out] : buffer for storing a string obtained n[in] : size of buffer for the string
Return value	1: success, 0: failure

```
long writeMaterialParameterInteger
(JcfDataPtr data, int id, const char* paramName, long value);
```

Description	Writes the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter value[in] : written value (integer)
Return value	1: success, 0: failure

```
long writeMaterialParameterDouble
(JcfDataPtr data, int id, const char* paramName, double value);
```

Description	Writes the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter value[in] : written value (floating point number)
Return value	1: success, 0: failure

```
long writeMaterialParameterComplex
(JcfDataPtr data, int id, const char* paramName, double real,
double img);
```

Description	Writes the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter real[in] : written value (real part of a complex number) img[in] : written value (imaginary part of a complex number)
Return value	1: success, 0: failure

```
long writeMaterialParameterString
(JcfDataPtr data, int id, const char* paramName, const char*
str);
```

Description	Writes the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter str[in] : written string
Return value	1: success, 0: failure

```
long readMaterialParameterIntegerByIndex
(JcfDataPtr data, int index, const char* paramName, long*
result);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in] : name of parameter result[out] : Obtained value (integer)
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readMaterialParameterDoubleByIndex
(JcfDataPtr data, int index, const char* paramName, double*
result);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management id[in] : ID of material paramName[in]: Parameter name result[out]: Obtained value (float)
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readMaterialParameterComplexByIndex
(JcfDataPtr data, int index, const char* paramName, double*
real, double* img);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management index[in]: Material ordinal number paramName[in] : Parameter name real[out]: Obtained value (real complex value) img[out] : Obtained value (imaginary complex value))
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long readMaterialParameterStringByIndex
(JcfDataPtr data, int index, const char* paramName, char* buff,
int n);
```

Description	Reads the parameters set for a material.
Parameters	data[in] : pointer indicating data for management index[in]: Material ordinal number paramName[in] : Parameter name buff[out]: Buffer for storing the obtained string n[in]: Buffer size
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeMaterialParameterIntegerByIndex
(JcfDataPtr data, int index, const char* paramName, long
value);
```

Description	Sets the parameters for a material.
Parameters	data[in] : pointer indicating data for management index[in]: Material ordinal number paramName[in]: Parameter name value[in]: Output value (integer)
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeMaterialParameterDoubleByIndex
(JcfDataPtr data, int index, const char* paramName, double
value);
```

Description	Sets the parameters for a material.
Parameters	data[in] : pointer indicating data for management index[in]: Material ordinal number paramName[in]: Parameter name value[in]: Output value (float)
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeMaterialParameterComplexByIndex
(JcfDataPtr data, int index, const char* paramName, double
real, double img);
```

Description	Sets the parameters for a material.
Parameters	data[in] : pointer indicating data for management index[in] : Material ordinal number paramName[in] : Parameter name real[in] : Output value (real complex value) img[in] : Output value (imaginary complex value)
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

```
long writeMaterialParameterStringByIndex
(JcfDataPtr data, int index, const char* paramName, const char*
str);
```

Description	Sets the parameters for a material.
Parameters	data[in] : pointer indicating data for management index[in] : Material ordinal number paramName[in]: Parameter name str[in] : String to be output
Return value	1: success, 0: failure
Remarks	Ordinal numbers (index) and IDs are different. IDs may not be consecutive missing some numbers in the middle, whereas ordinal numbers are always consecutive.

15-4. Using the JCF File Access Library (SimpleFHI)

15-4-1. Using C Language

15-4-1-1. Building/Executing a Sample Using VisualC++

Description will be given below on the assumption that the computer has Microsoft VisualC++ version 6.0 installed on Microsoft Windows 2000/XP.

1. Set Path for the environment parameter of Windows.

i. Select [Control Panel] from the Windows [start] menu (in the case of Windows XP).

ii. Select the "System" button in Control Panel by double-clicking it.

The [System Properties] dialog box will appear.

iii. Click the [Advanced] tab, and then click [Environment Variables].

The [Environment Variables] dialog box will appear.

iv. Select "Path" parameter in the list, and then click [Edit].

The [User Edition Parameters] dialog box will appear.

v. Into the [Parameters Value] text box, enter the installation directory of JMAG.

2. Start VisualC++, version 6.0 (VC hereinafter).

3. Set an included path of VC.

i. Select [Tools] > [Options] from the VC menu bar.

The [Options] dialog box will appear.

ii. Click the [Directories] tab, and then select "Include files" from the [Show Directories for] combo box.

iii. Add \(JMAG installation directory)\Tools\SimpleFHI to the [Directories] list box.

4. Set library path.

i. Select [Tools] > [Options] from the VC menu bar.

The [Options] dialog box will appear.

ii. Click the [Directories] tab, and then select "Library files" from the [Show Directories for] combo box.

iii. Add \(JMAG installation directory)\Tools\SimpleFHI to the [Directories] list box.

5. Read the sample project file.

The location for storing the project file is as follows.

```
(JMAG Installation
directory)\Tools\SimpleFHI\samples\C\Sample.dsp
```

To prepare a new project, it is necessary to perform the following settings (the settings have been implemented in the case of Sample.dsp described above).

i. Select [Project] > [Settings] from the VC menu bar.

The [Project Settings] dialog box will appear.

ii. Click the [C/C++] tab.

iii. Select [Generation of Code] from the [Category] combo box, and [Multi-Thread (DLL)] from the [Use run-time library] combo box.

iv. Click the [Link] tab.

v. Select "General" in the [Category] combo box, and then enter "Simple-FHI.lib" into the [Object/library modules] text box.

6. Build the file.

i. Select [Build (B)] > [Build (B)] from the menu bar.

Sample.exe is prepared.

7. Execute Sample.exe.

Open the command prompt, and then execute the following command.

```
sample input_jcf_file_name output_jcf_file_name
```

15-4-1-2. Building/Executing a Sample Using Visual Studio.Net

1. **Set Path for the environment parameter of Windows.**
   i. **Select [Control Panel] from the Windows Start menu (in the case of Windows XP).**
   ii. **Select the "System" button in Control Panel by double-clicking it.**
   The [System Properties] dialog box will appear.
   iii. **Click the [Advanced] tab, and then click [Environment Variables].**
   The [Environment Variables] dialog box will appear.
   iv. **Select "Path" parameter in the list, and then click [Edit].**
   The [User Edition Parameters] dialog box will appear.
   v. **Into the [Parameters Value] text box, enter the installation directory of JMAG.**
2. **Start Visual Studio.Net 2003 (VS hereinafter).**
3. **Set an included path of VS.**
   i. **Select [Tools] > [Options] from the VS menu bar.**
   The [Options] dialog box will appear.
   ii. **Click the [Directories] tab, and then select "Include files" from the [Show Directories] combo box.**
   iii. **Add \(JMAG installation directory)\Tools\SimpleFHI to the [Directories] list box.**
4. **Set library path.**
   i. **Select [Tools] > [Options] from the VS menu bar.**
   The [Options] dialog box will appear.
   ii. **Select [Project] > [VC++ Directories] via the tree.**
   iii. **Select "Library files" from the [Show Directories for] combo box.**
   iv. **Add \(JMAG installation directory)\Tools\SimpleFHI to the list box.**

5. Read the sample project file.

The location for storing the project file is as follows.

```
(JMAG installation
directory)\Tools\SimpleFHI\samples\C\Sample.vcproj
```

To prepare a new project, it is necessary to perform the following settings (the settings have been implemented in the case of the Sample.vcproj described above).

i. Select (Project Name) using the solution explorer.

ii. Select [Project] > [Properties] from the VS menu bar.

The [(name of project) Property Page] dialog box will appear.

iii. Select [Configuration Properties] > [Linker] > [Input] via the tree.

iv. Add SimpleFHI.lib to the [Additional Dependencies] cell.

v. Select [Configuration Properties] > [C/C++] > [Precompiled Headers] via tree.

vi. Select [Not Using Precompiled Headers] in the [Create/Use Precompiled Header] cell.

6. Build file.

i. Select [Build] > [Build Sample] from the menu bar.

ii. Sample.exe is prepared.

7. Execute Sample.exe.

Open the command prompt, and then execute the following command.

```
sample input_jcf_file_name output_jcf_file_name
```

15-4-1-3. Building/Executing a Sample Code Using Linux

1. **Move to the directory where the sample code is located.**

   ```
   (JMAG Installation directory) /solver/mod/tools/SimpleFHI/
   samples/C/Sample
   ```

2. **Run the make command.**

   SimleFHI is generated.

3. **Specify the directory containing the JCF file handler-related files for the environmental variable LD_LIBRARY_PATH.**

   (Example of a command using bash: 32bit)

   ```
   LD_LIBRARY_PATH=(installation directory)/solver/mod/lib/
   linux32
   export LD_LIBRARY_PATH
   ```

4. **Run SimleFHI.**

   Run the next command. (Make sure to leave a space between the command and file name.)

   ```
   SimleFHI input_JCF_file_name output_JCF_file_name
   ```

15-4-2. Using Excel VBA

15-4-2-1. Using a Sample Macro

Description will be given below on the assumption that the computer has Microsoft Excel installed on Microsoft Windows 2000/XP.

1. Set path for the environment parameters of Windows.

i. Select [Control Panel] from the Windows [start] menu (in the case of Windows XP).

ii. Select the "System" button in Control Panel by double-clicking it.

The [System Properties] dialog box will appear.

iii. Click the [Advanced] tab, and then click [Environment Variables].

The [Environment Variables] dialog box will appear.

iv. Select "Path" parameter in the list, and then click [Edit].

The [User Edition Parameters] dialog box will appear.

v. Into the [Parameters Value] text box, enter the installation directory of JMAG.

2. Start Microsoft Excel (Excel hereinafter).

3. Change the macro security level.

i. Select [Tools] > [Options] from the Excel menu bar.

The [Options] dialog box will appear.

ii. Click the [Security] tab, and then click [Macro Security].

The [Security] dialog box will appear.

iii. Click the [Security Level] tab, and then select the [Intermediate] radio button.

Important **Changing the macro security level may increase the risk of making your machine susceptible to the attack of external rogue programs. This operation should be performed with utmost care in accordance with the security policy adopted by the company or group to which you belong.**

4. Read the sample file.

i. Open the sample at \(JMAG installation directory)\ Tools\SimpleFHI\samples\ExcelVBA\sample.xls.

ii. A message dialog will appear that reads "'...\sample.xls' contains macro." Click the [Activate Macro] button.

5. Execute the sample file.

i. Select the [JCF Editing Sample] sheet.

ii. Click the [Read...] button.

The [Open File] dialog box will appear.

iii. Select a desired JCF file, and then click [Open]. In the [Name of File to be Opened] cell, there appears the name of the selected file.

The values used for the setting of the file are read into the cell.

iv. Change the values in the cell as appropriate.

v. Click the [Write...] button.

The [Name File for Storage] dialog box will be displayed.

vi. Enter a name of the file, and then click [Storage].

6. Verify the source codes of ExcelVBA.

i. Select [Tools] > [Macro] > [Macro] from the Excel menu bar.

The [Macro] dialog box will appear.

ii. Click the [Edit] button.

To use the API of SimpleFHI in ExcelVBA, it is necessary to copy declare.txt and paste its contents to the file (sample.xls has necessary contents pasted in advance). The part in question in sample.xls is as seen below.

```
' Reference to the parameters provided by SimpleFHI is
declared.
' (begin)----------------------------------------

' Loading JCF file
Public Declare Function loadJcfFile Lib "SimpleFHI_32_10"
(ByVal path As String, ByRef ptr As Long) As Long
  :
(omitted)
  :
Public Declare Function writeMaterialParameterString Lib
"SimpleFHI_32_10" _
```

```
(ByVal ptr As Long, ByVal index As Long, ByVal paramName As
String, ByVal str As String) As Long

' (end)----------------------------------------
```

Reference

For the details of Excel VBA, see reference books commercially available.

Chapter 16. Extracting Data from a Result File (JMAG-Designer)

This chapter describes extracting data from a JMAG-Designer project file (*.jproj) or a result file (*.jplot) using the data extraction tool without running JMAG-Designer.

16-1. Function

Project files (*.jproj) or result files (*.jplot) for JMAG-Designer are described in binary format.

Therefore, this function is used when extracting result data from binary files.

This tool has the following characteristics.

- This tool is a console application.
- This tool can be used in either Windows or Linux environment.
- Specifies a result file (*.jplot) or a calculated project file (*.jproj) when extracting a result data.
- Specifies a project file (*.jproj) when extracting parameters for settings.
- Data for results or parameters for settings that are extracted are exported as standard output text files.

16-2. Important Notes Regarding the Use of This Function

- The following types of data for results can be exported.
  - Items that are exported as values for each step
- The following types of data for results cannot be extracted.
  - Items that have an amount of distribution
  - Items that require processing in JMAG-Designer such the probe setting
- A JCF file (*.jcf) and a result file (*.jplot) need to be exported to the Case folder under the jfiles folder when extracting result data or setting parameters from a project file (*.jproj).

  MEMO A jfiles folder is automatically created when a project file (*.jproj) is saved. For example, a folder is created in the following hierarchy structure when a JMAG-Designer project is saved in the D drive of the machine (project name: AAA, model name: BBB, study name: CCC)

  D:\AAA.jfiles\BBB~0\CCC~1\Case1

  The [AAA.jfiles] is for [jfiles].

  MEMO A JCF file (*.jcf) and a result file (*.jplot) is exported to a Case folder when running a calculation for a study.

16-3. Running Procedures (Windows/Linux)

The following are explanations on the operations for this tool.

- Extracting result data
- Obtains list of output items
- Obtains a list for a model or a study included in the project
- Obtains value for each parameter set in a project

16-3-1. Important Notes Regarding the Descriptions of Commands

The following should be noted when using this function.

- File paths and query needs to be enclosed in double quotation marks (“).
- Command lines and arguments needs to be defined in one line.
  A conjunction [^] symbol needs to be placed after a line break when dividing a command line into two lines in a Windows environment. A backslash [\] symbol needs to be placed after a line break when dividing a command line into two lines in a Linux environment.

16-3-2. Extracting Result Data

A result data is extracted by specifying a result file (*.jplot) or a project file (*.jproj).

1. Specify the following command.

■ Extracting result files (*.jplot)

```
“(JMAG installation directory)\jquery.exe” -file=”JPLOT file” -
query=”select (component) from /(output item)”
```

Example: Extracting the X, Y, and Z component of an electromagnetic force from jmag.jplot.

```
“C:\Program Files\JMAG-Designer12.0\jquery.exe” -
file=”D:\work\jmag.jplot” -query=”select X Y z from /Force”
```

■ Extracting from a project file (*.jproj)

```
”(JMAG installation directory)\jquery.exe” -file=”JPLOJ file” -
query=”select (component) from /(Model name/Study name/Case
name/Output items)”
```

Example: Extracting the X, Y, and Z component of a model AAA/study BBB/Case1 from jmag.jploj.

```
“C:\Program Files\JMAG-Designer12.0\jquery.exe” -
file=”D:\work\jmag.jproj” -query=”select X Y z from modelAAA/
studyBBB/case1/Force”
```

- The result data is exported as standard output in default settings. Use the options that are described below when exporting a file.
- Specify the model name, study name, and Case name when extracting result data from a project file (*.jproj).
- For more information about names for output items and components, see the following table.

■ Options

The following items can be specified as an option. The following default values in the table are used if an option is not specified.

Property name	Setting	Default	Description
-out	String		Specify the path for file export. Specify absolute path.
-format	"plain" "csv"	plain	Specify the format for file export. Plain text format CSV format

■ Output Items and component list

The output items that can be extracted are as follows.

Output items	Component	Description	Related analysis type
JouleLoss	Loss	Joule loss	Magnetic field analysis
HysteresisLoss	Loss	Hysteresis loss	Magnetic field analysis
Force	• X • Y • Z • Torque	Electromagnetic force (X, Y, Z), Torque	Magnetic field analysis, electric field analysis
LorentzForce	• X • Y • Z	Lorentz force (X, Y, Z)	Magnetic field analysis
Flux	Flux	Magnetic flux	Magnetic field analysis (static analysis, transient response analysis)
FluxReal	Flux	Magnetic flux (real)	Magnetic field analysis (frequency response analysis)
FluxImaginary	Flux	Magnetic flux (imaginary)	Magnetic field analysis (frequency response analysis)

Output items	Component	Description	Related analysis type
TerminalVoltage	Voltage	Circuit Voltage (real)	Magnetic field analysis (transient response analysis, frequency response analysis)
TerminalVoltageImaginary	Voltage	Circuit Voltage (imaginary)	Magnetic field analysis (frequency response analysis)
LineCurrent	Current	Current (real)	Magnetic field analysis (transient response analysis, frequency response analysis), electric field analysis
LineCurrentImaginary	Current	Current (imaginary)	Magnetic field analysis (frequency response analysis)
StoredEnergy	Energy	Stored energy	Magnetic field analysis
Inductance	Inductance	Winding inductance	Transformer analysis
DcResistance	Resistance	Winding DC resistance	Transformer analysis
CoilFlux	Flux	Flux linkage in coil	Transformer analysis
LineOnOff	Switch	ON/OFF information for circuit switch component	Magnetic field analysis
CurrentFlux	• Real • Imaginary	Magnetic flux for current condition (real, imaginary)	Magnetic field analysis
CoilFlux	• Real • Imaginary	Magnetic flux for FEM coil condition (real, imaginary)	Magnetic field analysis
CurrentCurrent	• Real • Imaginary	Current for current condition (real, imaginary)	Magnetic field analysis
CoilCurrent	• Real • Imaginary	Current for FEM coil condition (real, imaginary)	Magnetic field analysis

Output items	Component	Description	Related analysis type
Displacement	• X • Y • Z • Angle	Total displacement (X,Y,Z, rotation angle)	Magnetic field analysis, thermal analysis
BrushAngle	Angle	Contact angle for Brush motor and Commutator components	Magnetic field analysis
SurfaceCharge	Charge	Surface charge	Electric field analysis (static analysis, current distribution analysis)
SurfaceChargeReal	Charge	Surface charge (real)	Electric field analysis (frequency response analysis)
SurfaceChargeImaginary	Charge	Surface charge (imaginary)	Electric field analysis (frequency response analysis)
IronIronloss	• X • Y • Z	Iron loss (each component)	Iron loss analysis
IronHysteresisLoss	• X • Y • Z	Hysteresis loss (each component)	Iron loss analysis
IronJouleLoss	• X • Y • Z	Joule loss (each component)	Iron loss analysis
HeatFlow	Heat	Heat flow	Thermal analysis
HeatGeneration	Heat	Heat source	Magnetic field analysis, thermal analysis
AverageTemperature	Temperature	Average temperature	Thermal analysis
TerminalTemperature	Temperature	Terminal temperature	Thermal analysis

16-3-3. Obtaining List of Output Items

A list of output items included in a result file (*.jplot) or a project file (*.jproj) is obtained by specifying the file.

1. Specify the following command.

■ Obtaining from result files (*.jplot)

```
"(JMAG installation directory)\jquery.exe" -file="JPLOT file" -
query="describe"
```

■ Obtaining from project files (*.jproj) by specification

```
"(JMAG installation directory)\jquery.exe" -file="JPROJ file" -
query="describe /(Model name/Study name/Case name)"
```

16-3-4. Obtaining a List for a Model or a Study Included in the Project

A list of models or studies that is included in a project file (*.jproj) can be obtained by specifying the file.

1. Specify the following command.

■ Obtaining a list of models

```
"(JMAG installation directory)\jquery.exe" -file="JPLOJ file" -
query="select name from /"
```

■ Obtaining a list of studies

```
"(JMAG installation directory)\jquery.exe" -file="JPLOJ file" -
query="select name from /(model name)"
```

MEMO Some of the data in the list for models or studies may be different depending on the version of JMAG-Designer the project file is saved in.

16-3-5. Obtaining Values Set in Each Parameter of a Project

The values set in each parameter of an analysis model can be obtained by specifying a project file (*.jproj).

1. Specify the following command.

```
"(JMAG installation directory)\jquery.exe" -file="JPROJ file" -
query="select (parameter name and number) from /(Model name/
Study name/Case name)"
```

Example: Obtaining values for stack length and step divisions from model AAA/study BBB/Case1 of jmag.jproj.

```
"C:\Program Files\JMAG-Designer12.0\jquery.exe" -
file="D:\work\jmag.jproj" -query="select Control-1/
model_thickness Step-1/step_division from modelAAA/studyBBB/
case1"
```

MEMO Always specify a model name or study name.

MEMO The values for entire cases that are included in a study can be obtained by omitting the [Case name] from the command.
Example: Obtaining the value of stack length from model AAA/study BBB/entire cases of jmag.jproj.

```
"C:\Program Files\JMAG-Designer12.0\jquery.exe" -
file="D:\work\jmag.jproj" -query="select Control-1/
model_thickness from modelAAA/studyBBB"
```

MEMO Specify the block name and property name that is described in a JCF file as a parameter name.

▶▶ Reference
For more information about block names and property names for JCF files, see "JCF/PLOT File Format."

INDEX

Symbols

.clb 942
.iron 893
.jcf 781, 786, 791, 795, 802, 807, 810, 813, 815, 817, 886, 892
.jplot 782, 786, 816, 968, 969
.loss 814, 951
.msv 764
.nas 975
.pa 1055, 1067, 1116, 1121
.plot 782, 786, 816
.rtt 1054, 1067, 1072, 1081, 1085, 1089, 1097, 1100, 1103, 1115, 1121, 1125, 1133, 1144, 1190
.uni 968
.voltage 724
.xml 1067, 1085, 1100, 1121
$subite 625
$subtol 625

Numerics

4-sided Solid Element 652
5-sided Solid Element 653
6-sided Solid Element 651

A

A Method 310
Abaqus 823
Absorption Direction 709
Absorption Parameter 710
acce 635
Acceleration (output item) 617, 698
Acceleration Coefficient 307, 461, 534, 736
Acceleration Force 696
Acceleration Load 589
Acceptable Error Range 765
Accuracy Type 1056
ACOUST 682
Acoustic Analysis 682
Adaptive Mesh Control 762
Adaptive Method 737
Adding
    data to the JMAG-RT model 1028
Additional Force 197
Adhesion 604, 680
ADTEMP 952
Air (material properties) 61
Aisotropy 41
Allow Eddy Current 99
Allowing
    eddy currents 99
    electric insulation 99
AMG Method 735
Amount of Charge (output item) 570
Amount of Heat Source (output item) 497
Amplitude/Phase 979
Analysis Control 733
Analysis Control File Format 944
Analysis Error (output item) 759
Analysis Result Folder 988
Analysis Type 269, 457, 529
Angle Speed 192
Angle_DisplayFlag 1070, 1087, 1101, 1123
Anisotropic Magnetic Steel Sheets 62
Anisotropic Material 2D 645
Anisotropic Material 3D 647
Anisotropy by Element 88

Anisotropy for the Electrical Conductivity of Superconductors 66
Antiperiodic Parallel 287
A-phi Method 1 310
A-phi Method 2 310
Apply Stress Analysis Result 897
Area Losses 789, 794, 799
ASPRING 680
Attenuation Coefficient 196
Average Loss 790, 833
Average of Rotation 951
Average Temperature (condition) (output item) 497
Average Temperature (output item) 497
Average Temperature Calculation 450
Axial Reverse Rotational Periodic Boundary 106
Axis Direction Pattern (circular direction) 82

B

Back emf Flag 1070, 1087, 1101, 1123
Background 862
Base Distance 935, 940
Base of Temperature Flag 952
Basic Frequency 788, 890, 900
Basic Information 1039, 1075, 1091, 1105, 1128, 1141
Beam 577
Beam Element 656
Beam Element Properties 641
Bean Model (superconductivity properties) 64
BEMF_DisplayFlag 1070, 1087, 1101, 1123
B-H curve 16, 19, 22, 25
B-H curve (stress-dependent) 23, 29
B-H curve (temperature-dependent) 24, 30
B-M curve 22, 25
B-mu curve 22, 25
bmuser 562
Boltzmann constant 435
Brush/Commutator component 172

C

Calculate Coil Inductance 988
Calculate Far Field 947
Calculate Field Value 939
Calculate from Poles and Revolution Speed 890
Calculate Inductance with Saturation Flux 989
Calculate Iron Loss 886, 887
Calculate Local Progress Bar 1024
Calculate Motor Inductance 979
Calculate Nodal Force Accurately 766
Calculate Particle Trajectory 934
Calculation Method 270
Calculation Method for Nonlinear Iteration 312
Calculation Mode 1031, 1046, 1079, 1095, 1110, 1131, 1143
Calculation Resolution 1200
Capacitor component 159
CBAR 656
CEND 626
Centrifugal Force 590, 663
CG Method 735
CH 500
Charge Density (output item) 699
Charge Distribution Analysis 500, 546
CHEXA 651
China Steel 47
Circle Cross Section 578
Circuit 147
  circuit component types and items to be set 180
Circuit Conversion 286
Circular Direction Anisotropy 84
cndusr 32, 543
Coaxial Cable 716

coef_coil temperature 1060
Coefficient/Multiplier File 881
coef_flux 1059, 1069, 1118, 1122
coef_force 1118, 1122
coef_inductance 1059, 1069, 1118, 1122
coef_inertia1 1069, 1086, 1122
coef_inertia2 1069, 1086, 1122
coef_magnet 1059, 1069, 1118, 1123
coef_magnet temperature 1060
coef_mass1 1101
coef_mass2 1101
coef_material 1059, 1069, 1118, 1123
coeftable_torque_flag 1070, 1123
coef_torque 1059, 1069
Coil (material properties) 61
Coil Temperature 1060
Coil Temperature Correction 1200
Comment 623, 1040, 1076, 1091, 1106, 1129, 1142
Complete Magnetization (Isotropic/Anisotropic) 860, 865
Complex Correction 1050, 1051, 1112, 1113
Concentrated Load 586
Concentrated Mass 607, 677
Condition to Finish Analysis 936
Conductor 521
configuration factor 435
Configuration of FEM Coil 1041, 1076, 1092, 1106, 1129, 1142
CONM2 677
Connection Type 1059, 1118
Constant Temperature Boundary 428
Constraint 600
Contact Thermal Resistance 452
Control Data of Charge Distribution Analysis 526
Control File Size 319, 465
Control Output Interval 320, 465
Control Output Interval Table 320
Control Output Items 635, 739
Control Output Materials 319
Control Output Parts 319, 465
conusr 31, 349
Convergence Tolerance 307, 312, 461, 463, 534, 536, 736
Convert to 3D 967
Convert to Full Model 968
Coordinate System 59
Coordinate System Conversion 895
Copy Subcircuit File 1027
Copy to 1026
CORD2C 669
CORD2R 668
Correcting
  B-H Curve for frequency response analysis 315
  magnetizing properties 93, 98
Correction Coefficients 1200
Correction Multiplier 192, 197, 198
Coupled Analysis 769, 780, 787, 814
CPENTA 653
CPU Group Name 991
CQUAD4 654
Create 1076, 1092, 1106, 1129, 1142
Create Model 1024
Creating
  a model for JMAG-RT using the command Line 1149
  JMAG-RT Model from user prepared data 1033
Creating Averaged Loss 949
Critical Current Density 66
CTETRA 652
CTRIA3 655
Current 113, 1076, 1130
  current over a periodic boundary 119
Current (Ld, Lq) 1042
Current (output item) 411, 569, 759
Current (Spatial Harmonics) 1043
Current Calculation 519, 723

Current Density 125
Current Density (output item) 411, 569, 759
Current Density Boundary 516
Current Distribution Analysis 499
Current Flag 1070, 1087, 1102, 1124
Current Phase 979
Current Probe component 179
Current Signal Input 1063, 1084, 1099, 1120, 1138, 1148
Current Source (1 Terminal) component 168
Current Threshold 247
Cycle Type 737
Cylindrical Coaxial Cable 718
Cylindrical Coordinate System 60, 669, 941
Cylindrical Wave Guide 716, 717

D

DAIDO ELECTRONICS 47, 49
Daido Steel 49
Damping Coefficient ID 629
DAREA 659
Data File List 1026
d-axis Current/q-axis Current 980
Debugging 405
Delete result file 1047, 1080, 1096, 1111, 1132, 1143
demagnetization 12
demagnetizing field 13, 14
Demagnetizing Ratio (output item) 413
Density 38
Design Variable File 1011
Differential Permeability (output item) 413
Diode component 162
Direct Correction 1050, 1112
Direct Method 270, 734
Displacement 256, 277, 596
Displacement (output item) 412, 497, 617, 698
Displacement Coordinate System 670
Displacement Current 101, 305
Displacement of the Mover 197
Displacement per Period of Electrical Angle 1106
Displacement Threshold 248
DispLimitMax 1101
DispLimitMin 1101
Distributed Calculation 1046, 1079, 1095, 1110, 1131, 1143
Distributed Calculation (Remote) 980, 991
Distributed Memory Multiprocessor 297
Divergence Tolerance 309
Division Method 1001
Divisions of Time 774
DLL of Magnetic Field Analysis 390
DLOAD 630, 657, 975
DMP 297
Do not leave result files 1047, 1080, 1096, 1111, 1132, 1143
Do Not Output Distribution to Result File 1047, 1080, 1095, 1111, 1131, 1143
DP 70
DPHASE 660
Drive Type 1107
dtuser 473, 558
Dynamic Load 658
Dynamic Load ID 630, 657

E

Eccentricity 200
Eddy Current 99, 124, 129
Eddy Current Loss 905
Eddy Current Loss Density 905
Edge Force 694
Efficiency Map 1187, 1199, 1203
Eigen Value 612, 637
Eigen Value ID 628
Eigenmode Analysis 573
EIGRL 637

EL 499
ELCRD 690
Electric Charge (output item) 570
Electric Conductivity 32, 33
Electric Field (output item) 411, 569, 759
Electric Field Analysis 499
Electric Potential (output item) 412, 569, 617, 698
Electric Potential Boundary (CH) 513
Electric Potential Boundary (EL) 511
Electric Potential Probe component 179
Electric Potential Source (1 Terminal) component 165
Electric Potential Source (2 Terminals) component 171
Electric Power (output item) 412, 415, 416
Electric Properties 31
Electric Properties (temperature-dependent) 33
Electric Resistivity 31, 32, 33
Electric Signal Input 1061, 1083, 1099, 1119, 1136, 1147
Electrical Conductivity 31, 703
Electrode 690
Electromagnetic Force 591, 683
Electromagnetic Force (output item) 412
Electromagnetic Force and Torque Calculation 216
Electromagnetic Wave Frequency Response Analysis Module 701
Elevation from X-axis 86, 87
Enable the Setting of Resolution 1077, 1108
End 697
End of Control Section 626
End Time to Apply 774
ENDDATA 697
Endotherm Coefficient 423
Enlargement Distance 851
Environment Variables 242, 295, 408, 409, 837, 1216, 1247, 1249, 1252
epsusr 753
Equation of Motion 189
Equation of Motion (PSIM only) 1048, 1078, 1093, 1109
Equations for Iron Loss Calculations 906
Equilibrium Temperature 423
Equivalent Current Calculation 758
Equivalent Surface Current File 947
Excel VBA 1252
Excitation Mode 715
Execute FEM Calculation 1022
Executing
  analysis using third-party job management systems 1184
Expand from Half Cycle 968
Exporting Multi-purpose Files 969
Extended Distance 827
Extended to 3D Model 973
Extended to the Full Model the Partial Model 973
Extension of the Spring 190, 192, 195, 197
External Current 129
External Field 260, 719
External Field (Line Current) 720
extusr 745

F

Face Charge 523
Face Charge Density (output item) 571
Face Pressure (output item) 618
Factor 1057, 1082, 1098, 1117, 1135, 1145
Falling Angle 79, 80, 83
Far Field Calculation 756
FEM 270
FEM + BEM 205, 270
FEM Coil 131
FEM Coil component 151
FEM Conductor 141
FEM Conductor component 153
FFT 875

FFT Resolution 827
File Name of Magnetic Analysis Result 949
Final Frequency 1001
Find by Suffix 939
First Frequency 1001
Fixed Temperature component 447
flgph 483
Flux Linkage 123, 136
Flux Linkage in Coil (output item) 413
Flux of FEM Coil (output item) 411
Flux Weakening 1211
FORCE 662, 975
Force 216, 662
Force by Electric Field (output item) 569
Force Calculation 216
Force Calculation by Electric Field 517
Force Correction Point Sequence File 1116, 1121
Force Export for Virtual.Lab 966
Force Export Tool for Virtual.Lab 966
Force Flag 1102, 1124
Force for Structural Analysis 956
Force ID 632
Force Mapping Tool for Abaqus 826
formula
  calculating the demagnetizing ratio 414
  calculating the magnetic flux in the FEM coil and the magnetic flux with the current condition 414
  electric power 415, 416
  mises stress 618, 699
  sound pressure level 618, 699
FQ 71
FREQ 627
FREQ1 636
Frequency 731
Frequency Control 530, 613
Frequency ID 627
Frequency Interval 279, 614
Frequency Point Sequence 280, 615
Frequency Range 636
Frequency Response Analysis 71, 499, 573
Frequency Scale Factor 900
Frequency Separation Method 922
Friction Coefficient 192, 198
Friction Factor 1057, 1082, 1098, 1117, 1134, 1145
Friction Force 198
Friction Torque 193
Frozen Permeability 251
Full Model Conversion 283
Full Model Extension 805
Function Type 990

G

Gap 262
Gap Element 68
GCR method 735
Generating
  mesh with the motion condition set 205
Global Strain (output item) 618
GMG Method 735
GMRES Method 736
GRAV 696
GRID 650
Grobal Stress (output item) 617
Ground component 179
GSPRING 674

H

Heat Capacitor component 446
Heat Flow (output item) 497
Heat Flow Calculation 451
heat flux 435
Heat Flux (output item) 497
Heat Flux Boundary 438
Heat Generation Mapping Tool For Abaqus

832
Heat Source 441
Heat Source Amount 441
Heat Source Calculation 224
Heat Source component 446
Heat Source Density 441
Heat Source Density (output item) 497
Heat Source Density Coefficient 441
Heat Transfer Boundary 431
Heat Transfer component 447
High Speed Solver 314
Hitachi Metals 51
Hoganas AB 50
hsusr4 377
HT 419
Hybrid Method 735
Hysteresis Loss (output item) 412, 417, 759, 904
Hysteresis Loss Density (output item) 412, 759, 904

I

ICCG 310, 461, 533
id=0 1209
I_DisplayFlag 1070, 1087, 1102, 1124
In/Out Port 714
Incomplete (Anisotropic) 860
Incomplete (Isotropic) 860
Incomplete Magnetization (Anisotropic) 866
Incomplete Magnetization (Isotropic) 866
incusr 743
Inductance 1077
Induction Matrix 1001
Induction Motor 1139
Inductor component 159
Inertia Moment 191, 192
Information 1039, 1075, 1091, 1105, 1128, 1141
Initial Displacement 190, 192, 195, 197
Initial Magnetization 10
Initial magnetization for permanent magnet 318
Initial magnetization for permanent magnet (output item) 413
Initial Particle Density 528
Initial Position 185, 190, 192, 195, 935
Initial Temperature 439
Initial Velocity 935
Input/Output of JMAG-RT Model Block 1060, 1071, 1083, 1088, 1102, 1119, 1124, 1136, 1147
Insulation 99, 264, 522
Internal Charge 571
Inverter Rating 1199
Iron Loss 1046, 1143
Iron Loss (output item) 904
Iron Loss Calculation 226
Iron Loss Density (output item) 904
Iron Loss Expression 894
Iron Loss File 882
Iron Loss File (Stress-dependent) 884
iron_str 927
Irreversible 15
Isotropic Magnetic Material 61
Isotropic Magnetic Steel Sheets 62
Isotropic Material 643
Iterative Matrix Solver 736
Iterative Method 270, 734, 735
I-V Model (superconductivity properties) 65

J

jbdll.dll 390
JFE Steel 53
JLOAD 683
JMAG-RT 1015
JMAG-RT Library Manager 1021, 1038, 1075, 1091, 1105, 1128
JMAG-RT Model Block 1068, 1082, 1117

JMAG-RT Model Data File 1054, 1067, 1072, 1081, 1085, 1089, 1097, 1100, 1103, 1115, 1121, 1125, 1133, 1144, 1190
JMAG-RT Viewer 1187
JMAG_RT_2_mono_step 1083
JMAG_RT_2_mono_step.mdl 1081
JMAG_RT_2_mono_step_Iin 1084
JMAG_RT_2_mono_step_Iin.mdl 1081
JMAG_RT_2_mono_step_Vin 1084
JMAG_RT_2_mono_step_Vin.mdl 1081
JMAGRT_DLL.dll 1172
JMAG_RT_IM 1147
JMAG_RT_IM.mdl 1144
JMAG_RT_IM_Iin 1148
JMAG_RT_IM_Iin.mdl 1144
JMAG_RT_IM_Vin 1148
JMAG_RT_IM_Vin.mdl 1144
JMAG_RT_lsm 1119
JMAG_RT_lsm.mdl 1115
JMAG_RT_lsm_Iin 1120
JMAG_RT_lsm_Iin.mdl 1115
JMAG_RT_lsm_Vin 1120
JMAG_RT_lsm_Vin.mdl 1115
JMAG_RT_pmsm 1061
JMAG_RT_pmsm.mdl 1054
JMAG_RT_pmsm_Iin 1063
JMAG_RT_pmsm_Iin.mdl 1055
JMAG_RT_pmsm_Vin 1062
JMAG_RT_pmsm_Vin.mdl 1055
JMAG_RT_solenoid 1099
JMAG_RT_solenoid.mdl 1097
JMAG_RT_solenoid_Iin 1099
JMAG_RT_solenoid_Iin.mdl 1097
JMAG_RT_solenoid_Vin 1099
JMAG_RT_solenoid_Vin.mdl 1097
JMAG_RT_SR_3phase 1136
JMAG_RT_SR_3phase.mdl 1133
JMAG_RT_SR_3phase_lin 1138
JMAG_RT_SR_3phase_lin.mdl 1134
JMAG_RT_SR_3phase_Vin 1137
JMAG_RT_SR_3phase_Vin.mdl 1133
JMAG_RT_SR_4phase 1136
JMAG_RT_SR_4phase.mdl 1133
JMAG_RT_SR_4phase_lin 1138
JMAG_RT_SR_4phase_lin.mdl 1134
JMAG_RT_SR_4phase_Vin 1137
JMAG_RT_SR_4phase_Vin.mdl 1133
JMAG_RT_SR_5phase 1136
JMAG_RT_SR_5phase.mdl 1133
JMAG_RT_SR_5phase_lin 1138
JMAG_RT_SR_5phase_lin.mdl 1134
JMAG_RT_SR_5phase_Vin 1137
JMAG_RT_SR_5phase_Vin.mdl 1133
JMAG-Scheduler 786
JMAT2 645
JMAT9 647
Joule Loss (output item) 412, 759, 904
Joule Loss Density 792
Joule Loss Density (output item) 412, 759, 904
JSOL (Sample Material Data) 58

K

KIDO 933

L

Lamination of Isotropic Magnetic Steel Sheet 97
Laser Beam 726
Ld, Lq Model 1022, 1038
License Type 294, 467, 539
Line Charge Density (output item) 571
Line of Magnetic Flux (output item) 411
Linear 11
Linear (constant) 21
Linear (temperature dependent) 22
Linear Solenoid 1090

Linear Synchronous Motor 1104
LINE_FORCE 694
Linking
to an external circuit simulator 199, 223
to PSIM and MagCoupler 244
LOAD 632
Load Torque 191
Local Coordinate System 59, 941
Lorentz Force (output item) 412
Lorentz force density 318
Lorentz Force Density (output item) 412
Lorentz Force Method 220
Loss Averaging 784
Loss Generation Area 833
Lower Limit of the Current Value 1002
Lumped Constant Component 725

M

Macro component 179
MagCoupler 243
MagCoupler-DL 243
Magnet 63
Magnet Temperature 1060
Magnet Temperature Correction 1200
Magnetic Field (output item) 411, 759
Magnetic Flux (output item) 411, 415
Magnetic Flux Calculation 230
Magnetic Flux Density (output item) 411, 759
Magnetic Flux in Current Condition (output item) 411
Magnetic Properties of a Stress Dependency 29
Magnetic Properties of a Temperature Dependency 30
Magnetization (output item) 411
Magnetization Direction 78
Magnetization for Magnetic Field Analysis 859
Magnetization Ratio 860
Magnetization Ratio and Recoil Relative Permeability 860
Magnetization Type 865
Magnetize When Running Analysis 861
Magnetizing Properties 10
anisotropic magnetic steel sheet 27
magusr 14, 23, 345
Main Window 1192
Major Loop 925
Map View Window 1194
Mass 196, 198, 1098, 1101, 1117, 1122
MAT1 643
MAT2 645
MAT9 647
Material Database 9, 44, 893
Material Editor 6
Material Properties 703
MATLAB/Simulink 1018
MATP 686
Matrix Solver 734
Max. Efficiency 1212
Max. Power 1212
Max. Value from Specified Step Range 859
Maximum Analytical Error 765
Maximum Coercive Force 860
Maximum Level of Division 935, 940
Maximum Nonlinear Iterations 312
Maximum Number of Iterations 309, 461, 462, 534
Maximum Number of Nodes in Coarse Mesh 738
Maximum Number of Stages 765
Maximum Torque / Flux 1210
Mechanical Properties
anisotropy 40
isotropic 40
MechTimeConstant 1068, 1086, 1101, 1122
Memory Size 948, 951
Merge Loss Files 887
Mesh 761
METH 628

Microstrip Line 716, 718
Minor Loop 925
Mises Strain (output item) 618
Mises Stress (output item) 617, 618, 698, 699
MITSUBISHI STEEL 54
Modal Analysis 619
Modal Damping Table 638
Modal Frequency Response Analysis 619
Modal Port 716
Model Divisions 1142
Model Folder 1040, 1076, 1092, 1106, 1129, 1142
Model Type 1022
Moment of Force 1 1212
Moment of Inertia 1057, 1082, 1134, 1145
MomentOfInertia 1068, 1086
Motion 182, 448
Motion condition and electromagnetic force condition to link 248
Motion Range 1092
Motion Type 1059, 1118, 1146
Motor Characteristics 1052, 1114
Motor Control 1199, 1208
Motor Information 1039, 1075, 1091, 1105, 1128, 1141
Motor Model 1040
MPC 634, 672
Multigrid Method 735
Multigrid Refinement 704
Multipoint Constraint 672
Multipoint Constraint ID 634

N

Nastran file 975
Natural Boundary 108, 427, 510, 706
Natural Modes 715
Netlist XML File 1067, 1085, 1100, 1121
Newton-Raphson 312
NGRP 670
Nippon Steel 54
No expansion 968
Nodal Charge (output item) 571
Nodal Coordinates 650
Nodal Force 318, 808
Nodal Force (Input Load) (output item) 618, 699
Nodal Force (output item) 412, 571, 618, 699
Nodal Force Method 219
Nodal Temperature 664
Nonlinear (B-H curve (stress-dependent)) 23
Nonlinear (B-H curve (temperature-dependent)) 24
Nonlinear (B-H curve) 22
Nonlinear (B-M curve) 22
Nonlinear (B-mu curve) 22
Nonlinear (irreversible, demagnetizing field) 14
Nonlinear (irreversible, thermal demagnetization, demagnetizing field) 13
Nonlinear (reversible, temperature-dependent) 12
Nonlinear (reversible) 11
Nonlinear Calculation 311, 462, 535
Nonlinear Magnetizing Properties for a Soft Magnetic Material 25
Nonlinear Magnetizing Properties of a Magnet 16
Non-magnetic Material 61
Normal Force 192, 193, 198
Number of Divisions 979, 980, 1001
Number of Divisions along Phi-direction 948
Number of Divisions along Theta-direction 948
Number of Integration Points 935, 940
Number of Iterations 736
Number of Mesh Extended 3D Model 975
Number of Motor Poles 979
Number of Nonlinear Iterations 535
Number of Phases 1129
Number of Poles 788, 1040, 1076, 1129,

1142
Number of Steps 952

O

Offset Angle 1058, 1070, 1083, 1087, 1135
Offset Disp 1124
Offset Displacement 1117
One-way Coupled Analysis 769, 778, 801, 813, 821, 823, 824, 836
Only calculation which went wrong is re-performed 1003
Option Window 1193
Orthogonal Coordinate System 941
Output Control 423, 464, 616, 781, 787, 792, 793, 798, 803, 808, 811, 818
Output Intervals 320
Output Items 317, 411, 464, 569, 617, 1052, 1114
Output PSIM LUT File 980
Output Step Interval 935
Output Trajectory File 936

P

Parallel 287
Parallel Anisotropy 84
Parallel Computing 294, 297
Parallel Pattern (Circular Direction) 79
Parallel Pattern (Rectilinear) 86
Parameters for Eigen Value Calculation 625
Part Properties 577, 581, 583
Partial Inductance 267
Partial Model 235, 454
Particle Data 526
Particle Flight Region 505
Particle Number Density (output item) 571
Particle Trajectory 933
Patch Mesh 208
PBAR 641
Perfect Electric Boundary 705
Perfect Magnetic Boundary 706
Perfect Matching Layer Boundary 709
Periodic Boundary 102, 424, 506, 602, 711
Periodic Boundary Condition 940
Periodic Boundary/Anti-periodic Boundary 104, 425, 508, 713
Periodic Point Sequence 130, 161, 167, 170, 187
Periodicity 973
Permanent Magnet 10, 78
Permeance Coefficient (output item) 413, 416
permittivity matrix 35
Phase Angle (Dynamic Load) 660
phase1 485
phase2 489
Piezoelectric Constant 686
piezoelectric stress constant matrix 42
Pitch P 86, 87
Plane Rotation Motion 209
Plasma 728
Plasma Density (output item) 759
Platform 1026
PLOAD 692
plsusr 749
plsusr_ini 747
PML Boundary 709
Point Sequence Data
  current 120
  current density 130
  diode component 164
  electric potential source 167
Point sequence with regular intervals per section 276, 281
Polar Anisotropy Pattern (Circular Direction) 81
Polar Anisotropy Pattern (Rectilinear) 87
Polar Coordinate System 941
Polarization Vector (output item) 571

Position Flag 1101, 1123
Post Output 316, 537
Power Amplitude 715
Power Calculation 722
Power Consumption 814
Power Consumption (output item) 569
Power Consumption Density (output item) 569
Power source component 172
Power Source Component Control 293
Preprocessing Types 736
PRESS 676
Press Fit 595, 676
Pressure (output item) 698
Pressure Load 587, 692
Principal Strain (output item) 618
Principal Stress (output item) 617, 618, 698, 700
Programs in C 1020, 1072, 1089, 1103, 1125, 1170, 1172
PSHELL 640
PSIM 243, 249, 980, 1018, 1067, 1085, 1100, 1121
PSOLID 639

Q

Quadrilateral Shell Element 654
Quality Factor (output item) 759
quser 494
QVOL 975

R

RA 1101
RA- 1086
RA+ 1086
Radial Anisotropic 84
Radial Pattern (Circular Direction) 80
radiation coefficient 435
Radiation Heat Transfer Boundary 434
Range of Motion 190
Range of Rotation 195
Rated 1142
Rated Current 247
RB- 1086
RB+ 1086
RBE2 678
Recalculate Failed Cases 991
Recalculate with Increased Resolution 1031
Rectangular Coordinate System 668
Rectangular Cross Section 580
Rectangular Wave Guide 716, 717
Reference Coordinate System 956
Reference Correction 1050, 1051, 1112, 1113
Reference Data Type 859
Reference End Step 899
Reference Flux Density (output item) 905
Reference Interval 827
Reference Plot File 859
Reference Start Step 899
reference temperature 435
Referenced Components of Magnetic Flux Density 893, 896
Regular Intervals 275, 280, 615
Relative Permeability (output item) 413
Relaxation Factor 1 313
Relaxation Factor 2 313
Remesh per Stage 766
Remote Execute 1002
Resistance of coil 1057, 1082, 1098, 1117, 1134
Resistor component 156
Resolution 1040, 1076, 1092, 1106, 1129, 1142
Resolution (current) 1093
Resolution (position) 1092
Restart 310, 328, 470, 737
Reversible 15

Reversible Settings for a Permanent Magnet 15
Revolution Speed 890
RFORCE 663
rgtusr 556
Rigid Body 599
Rigid Body Element 678
Rising Angle 79, 80, 83
RLOAD1 658
Rotation Axis to Eccentricity 204
Rotation Speed 788, 1059, 1146
Rotational Periodic Boundary 105, 426, 509
Rotor Angle Flag 1070, 1087
RT Correction 1049, 1112
RT Data File 1056, 1082, 1098, 1117, 1134, 1145
RT_DLL.dll 1067, 1072, 1085, 1089, 1100, 1103, 1121, 1125
RT_simulink.dll 1054, 1081, 1097, 1115, 1133, 1144
RU 1068, 1122
Running
  simulation using C language program 1170
RV 1068, 1122
RW 1068, 1122

S

Sample Time 1057, 1082, 1098, 1117, 1135, 1146
SAR (specific absorption rate) (output item) 759
Save File Magnetic Analysis Result 980
Save result at each stage if adaptive mesh generation 766
Scale Factor 851
Scaling Factor 659, 827, 832, 833
SDAM 629
Select User Subroutine DLL 902
Series 287
Set by External Program 277
Set Each Material 892, 897
Set Frequency 891
Set to Existing File 862
SetCoefficientsManually 1059, 1118
setdns 560
Setting of Inductance Matrix 1001
setusr 867, 868
Shared Memory Multiprocessing 294, 467, 539
Shell 581
Shin-Etsu Chemical 55
Show Full Path 939
Show Status 862, 958
SIBC 100
SimpleFHI 1222, 1226
Simulink 240, 250, 390, 1054, 1081, 1097, 1115, 1133, 1144
Single Point Constraint 665
Single Point Constraint ID 631
Size of Memory 952
Skew 88
Skip End Time 774, 777
Slide 206, 212
Smoothing 737
SMP 294, 467, 539
Snapshot Calculation 325
Soft Magnetic Material 20, 97
SOL 624
Solid 583
Solid Element Properties 639
Solver Calculation Control 610
Solver Type 624
Sound Pressure 608
Sound Pressure (output item) 617, 698
Sound Pressure Level (output item) 617, 618, 698
Spatial Harmonic Model 1022, 1038
SPC 665
Specific Heat 37

Specified Step 859
Specify Step Range 968
Speed (output item) 413
Speed Flag 1070, 1087
Speed_DisplayFlag 1070, 1087, 1102, 1123
Spherical Anisotropic 85
Spring 606
Spring Constant 196, 1098, 1101
Spring Element 674
Square Cross Section 579
SR Motor 1126
SSOR Method 736
ST 70
Start Point 86, 87
Start R 86, 87
Start Time to Apply 774
Starting Position 79, 80, 81
Static Analysis 70, 499, 573
Steady-state Approximate Transient Analysis 301
Step 731
Step Control 458, 530
Step Specification 833
Steps from End Steps 968
Stored Energy (output item) 317, 413
stre 635
Stress 892
Stress (Cylindrical Coordinate System) (output item) 617, 698
Stress (output item) 698
Stress Distribution 253
Stress-dependent Material 29, 876
Strict Criteria for Convergence Tolerance 314
Structural Analysis 619
Structural Analysis Input 952
Structural Analysis Output 952
Subcycling 246
Subdivision at Intersection 935, 940
Successive Method 313
Sumitomo Metal 55
Sumitomo Metal Industries 55
Superconductivity Properties 64
Surface Charge (output item) 570, 571
Surface Charge Distribution Calculation 518
Surface Current Density (output item) 411
Surface Force Density (output item) 412
Surface Force Method 220
Surface Impedance Boundary 708
Surface Impedance Method 100
Surface Joule Loss Density (output item) 412
Switch component 159
Symmetry Boundary 108, 510, 705
Symmetry Boundary Condition 940, 947

T

TABDMP1 638
Table (Dynamic Load) 661
TABLED1 661
TDK 56
TEMP 664
TEMPERATURE 633
Temperature (output item) 497
Temperature Boundary 429
Temperature Correction 1049
Temperature Distribution 255, 525
Temperature Evaluation Section 789, 794, 799
Temperature for Structural Analysis 952
Temperature ID 633
Temperature Load 593
Temperature-dependent Material 30, 786, 795, 815
Terminal Temperature (output item) 497
Thermal Analysis 419
Thermal Analysis Model 832, 833
Thermal Analysis Output 952
Thermal Conductivity 36
thermal demagnetization 12, 13
Thermal Electromotive Force Loss (output

item) 569
Thermal Electromotive Force Loss Density (output item) 570
Thermal Resistor component 445
Three-phase Current Source component 178
Three-phase Electric Potential Source component 178
Three-phase PM Synchronous Motor 1036
Time dependency of frequency point sequence 282
Time interval 274
Time Interval Settings 773
Time Periodic Explicit Error Correction 303
Time point sequence 276
Time Unit 247
TLOAD 975
T-mu curve 22
Tokyo Ferrite 56
Torque 1077
Torque (output item) 412
Torque Calculation 216
Torque Correction 1055, 1067
Torque Flag 1070, 1087
Torque_DisplayFlag 1070, 1087, 1102, 1124
Torque-Speed curve 1188, 1203
Total Displacement (output item) 413, 497
Total Iron Loss of All Frequencies (FQ) 899
Total Loss (output item) 570
Total Loss Density (output item) 570
T-p curve 33
TR 70
Transformation Definition 861
Transformation Table 861, 867
Transient Response Analysis 70
Translation Speed 1118
Translational Motion 208
Translational Periodic Boundary 106, 426, 509, 713
Transmitted Power (output item) 759
Transparent Boundary 707
Triangular Shell Element 655
T-sigma curve 33
turns in coil2 1083, 1086, 1135
turns_coil1 1060, 1069, 1119, 1123
turns_coil2 1060, 1069, 1101, 1119, 1123
Two-phase Stepping Motor 1073
Two-way Coupled Analysis 284, 460, 532, 770, 783, 817, 823, 839

U

ucond 36, 478
udnsty 38, 493
Uniform Refinement Levels 737
Universal file 968
Updating
    JMAG-RT model data file 1034
Upper Limit Distance of Parts (m) 1002
Use Average Joule Loss 780
Use Different Mesh 820
Use different mesh between magnetic field and structure 804
Use different mesh for each analysis 789, 794, 799
Use Force Coefficient File 1118
Use GPU 300
Use Iron Loss Results 833
Use IronLoss 1056, 1145
Use Loss File 894
Use Mag Results 832
Use Magnetization Program 863
Use Multiple Results 940
Use Rotation Average 833
Use Structural Analysis Result File 812
Use the value at minimal residue 310
Use Torque Coefficient File 1059
User Component (Negative Integer) 860
User Component Model (superconductivity properties) 65
User Data 1022

User Defined 860, 867
User Subroutine 14, 199, 323, 330, 468, 471, 540, 546, 715, 740, 927
User Subroutine component 162
User Subroutine DLL Name 902
usesee 544
Using
  GPU 300
  high speed solver 314
  permanent magnet 10
  soft magnetic material 20
  strict criteria for convergence tolerance 314
uspeci 37, 480
usrcir 162, 364
usrdns 552
usrene 381
usrfle 367
usrfm2 192, 198, 359
usrfma 199
usrgap 564
usrhrd 476
usrhtc 475
usrhys 379
usrlos 369
usrmtx 371
usrplt 383
usrsc 385
usrsee 545
usrstp 366, 474
usrstr 29, 386
usrxv 368

V

V Cycle 737
VAC 57
Vector Potential (output item) 411
velo 635
Velocity (output item) 617, 698
Velocity Flag 1102, 1123
velusr 554
Virtual Movement 204
Virtual.Lab 966
Visual Studio .Net 1249
VisualC++ 1247
Voltage (output item) 759
Voltage Calculation 724
Voltage Signal Input 1062, 1084, 1099, 1120, 1137, 1148
Voltage/Current Excitation 721
Volume Average of Magnetic Flux Density 414
Volume Charge 524
Volume Force 695
Volume Load 588
VOLUME_FORCE 695

W

Winding DC Resistance (output item) 413
Winding Inductance (output item) 413
WV 701

Y

Y2-axis Display 618